USER'S MANUAL OF FORMWORKS PLUS FOR VECTOR5

by

Kyle Blosser
Serhan Guner
Frank J. Vecchio

May 2016

© Copyright by K. Blosser, S. Guner, F. J. Vecchio (2016)

TABLE OF CONTENTS

1. Introduction 4

2. Create A Structural Model 4

2.1 Define Job 6

2.2 Define Material Properties & Cross-Sections 8

2.3 Define and Mesh Structure 9

2.4 Create Support Restraints 11

2.5 Element Output List 11

2.6 Move Nodes 12

2.7 Reassign Material Types 12

2.8 Optional Advanced Features 13

2.8.1 Create Nodes 13

2.8.2 Node Copy 13

2.8.3 Create Frame Elements 14

3. Applying Loads 14

3.1 Apply Nodal Loads 14

3.2 Apply Support Displacements 15

3.3 Apply Gravity Loads 15

3.4 Apply Concrete Prestrains 16

3.5 Apply Impulse Forces 16

3.6 Apply Temperature Loads 17

3.7 Apply End Action Loads 17

3.8 Apply Concentrated Loads 18

3.9 Apply Uniformly Distributed Loads 19

3.10 Clear Load Case 20

4. Display Options 20

5. Performing an Analysis 22

5.1 Retrieving Output Data 23

5.2 Troubleshooting 24

5.3 Visualizing the Analysis Results 25

6. Toolbar Menus 26

6.1 File 26

6.2 Edit 26

6.3 View 26
6.4 Job 27
6.5 Structure 27
6.6 Load 27
6.7 Analysis 28
6.8 Window 28
6.9 Help 29
6.10 Toolbar Icons 29
7. Conclusion 29

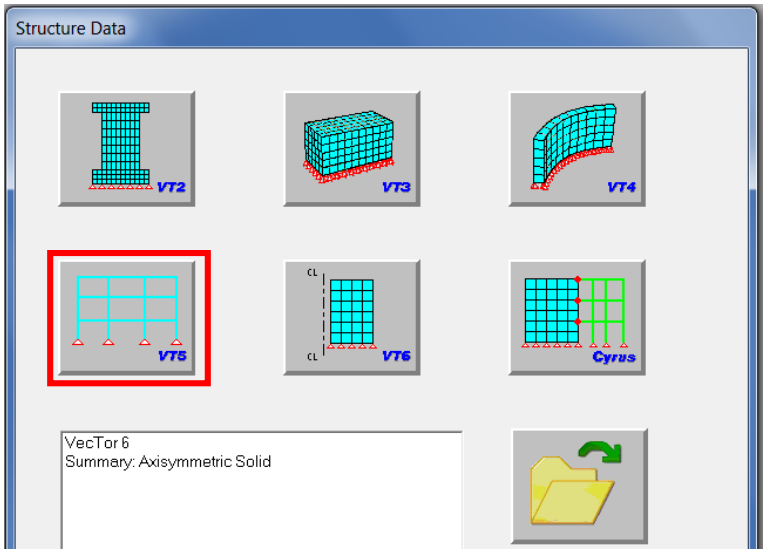
1. INTRODUCTION

FormWorks Plus prepares the text files required for a VecTor5 analysis, using a graphical interface. This document will allow a user to learn the FormWorks Plus program. Developing an understanding of this program will also allow the user to better understand the VecTor5 program and the input text files required. When a structural model is being created, FormWorks Plus will create the input text files required for the VecTor5 analysis as follows: Structure Data File (.S5R), Load Data Files (.L5R), Job Data File (.JOB), and the Auxiliary Data File (.AUX). All these files, the FormWorks model (.fwd), and the VT5.exe program should be placed in the same folder.

To better understand the VecTor5 program and the process behind the FormWorks Plus graphical interface, it is highly recommended to read the *User's Manual of VecTor5* and the *User's Manual of Janus for VecTor5*, both of which can be downloaded here. Watch and Learn videos can be found here. These videos show how to use the programs to model various structures.

2. CREATE A STRUCTURAL MODEL

When FormWorks Plus is opened, the following dialog box appears. This allows the user to select the type of structure that needs to be modeled. Select the icon for VecTor5 (VT5).

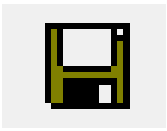


FormWorks Plus has the menus laid out in the taskbar so that the user can create and modify a structure. The taskbar is shown below:

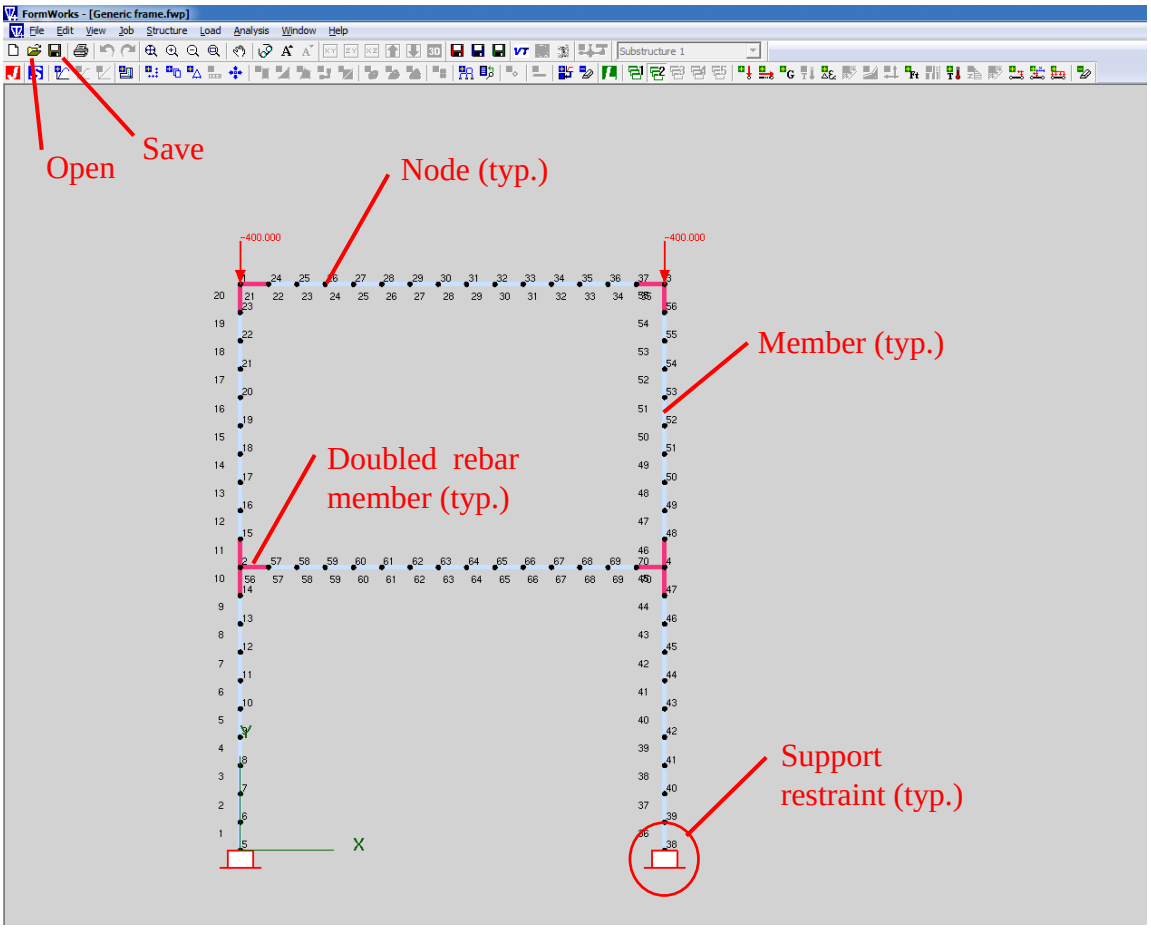
The menus that are necessary for structure creation are discussed in detail within this manual.

To load an existing structure, click the *Open* icon, shown below, which is located at the top left corner of the screen. Then, select the file with the ".fwp" extension to load the model.

To save the structure model, click on the *Save* icon. It is recommended that you save frequently.



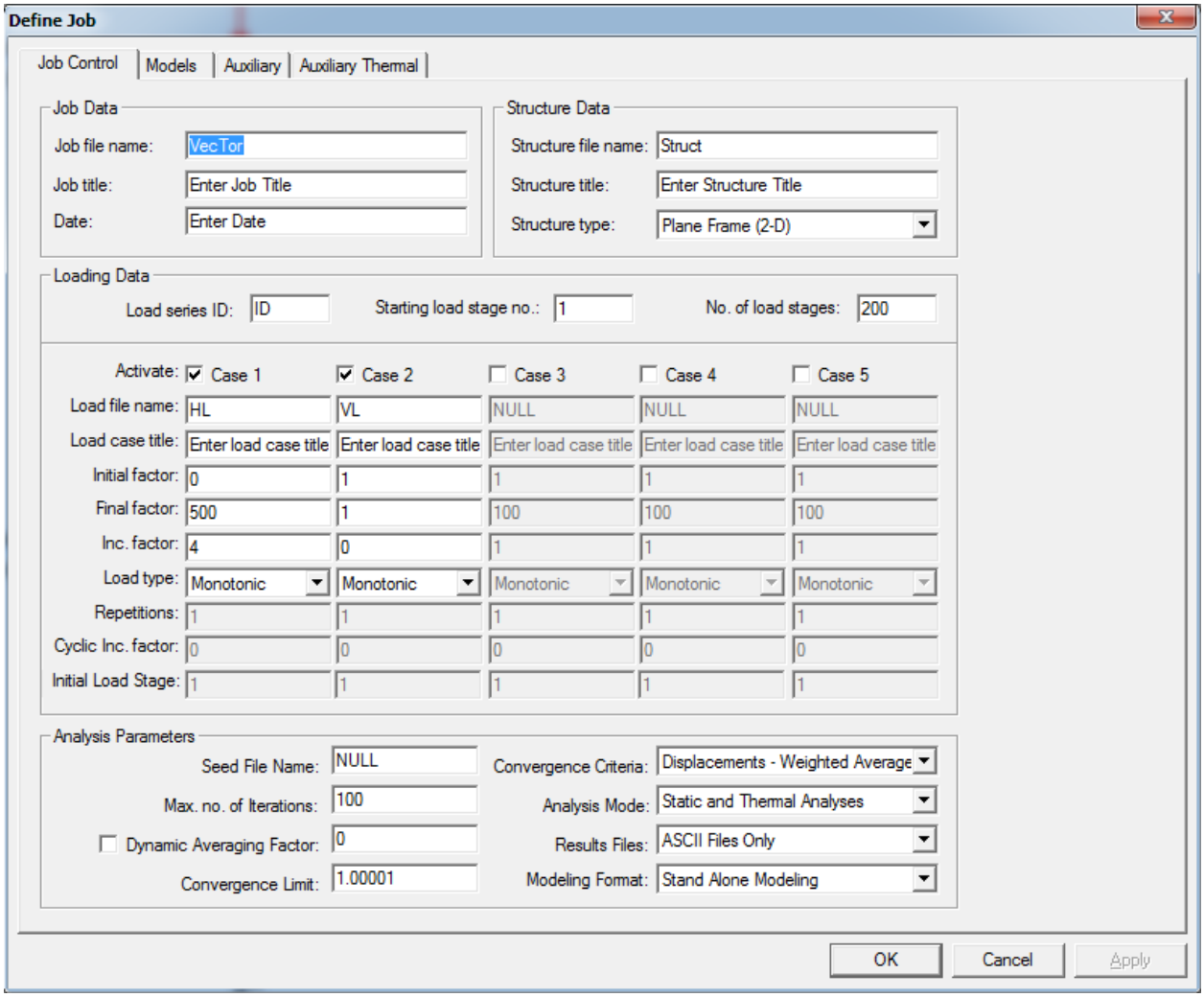
Upon opening a model, the screen should look something like the following image. Various parts of the structure are labeled in red annotation. Please note that if this is your first model with FormWorks Plus, you need to create it from scratch executing the tasks starting from Section 2.1.



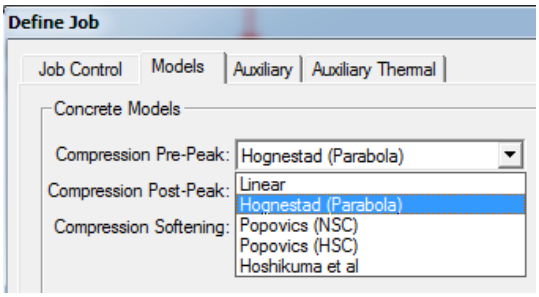
The frame above is shown with the members and node numbers laid out. The axial loads are shown for the load case 1, as 400 kN in the negative Y-direction, applied to nodes 1 and 3. The support restraints are also shown at the base of the structure, which are restraining nodes 5 and 38 in the X, Y, and Z-degrees of freedom. The red joint regions signify that semi-rigid end offsets were defined at the joint connections, which is important to remember during meshing.

2.1 Define Job

The first icon displays the *Define Job* menu. In this menu, the Job information is input. This information includes the number of load stages, type of loading, and loading increments. The load cases are also defined here. There are five possible load cases that can be applied to a single structure. At the top of the menu, the 'Job Control' tab allows the user to select the type of analysis calculations that are performed. These are explained in more detail in the *User's Manual for VecTor5*. Note that it is no more accurate for more load stages to be defined; just use enough so that the structure fails. The increment factor should be used to determine the increments for the loading. The following image shows an example structure that has been modeled using FormWorks Plus. Note the increasing load in Load Case 1 and the constant load in Load Case 2. The Load file name (in this example, HL for the horizontal load and VL for the vertical load) can be specified by the user as desired.

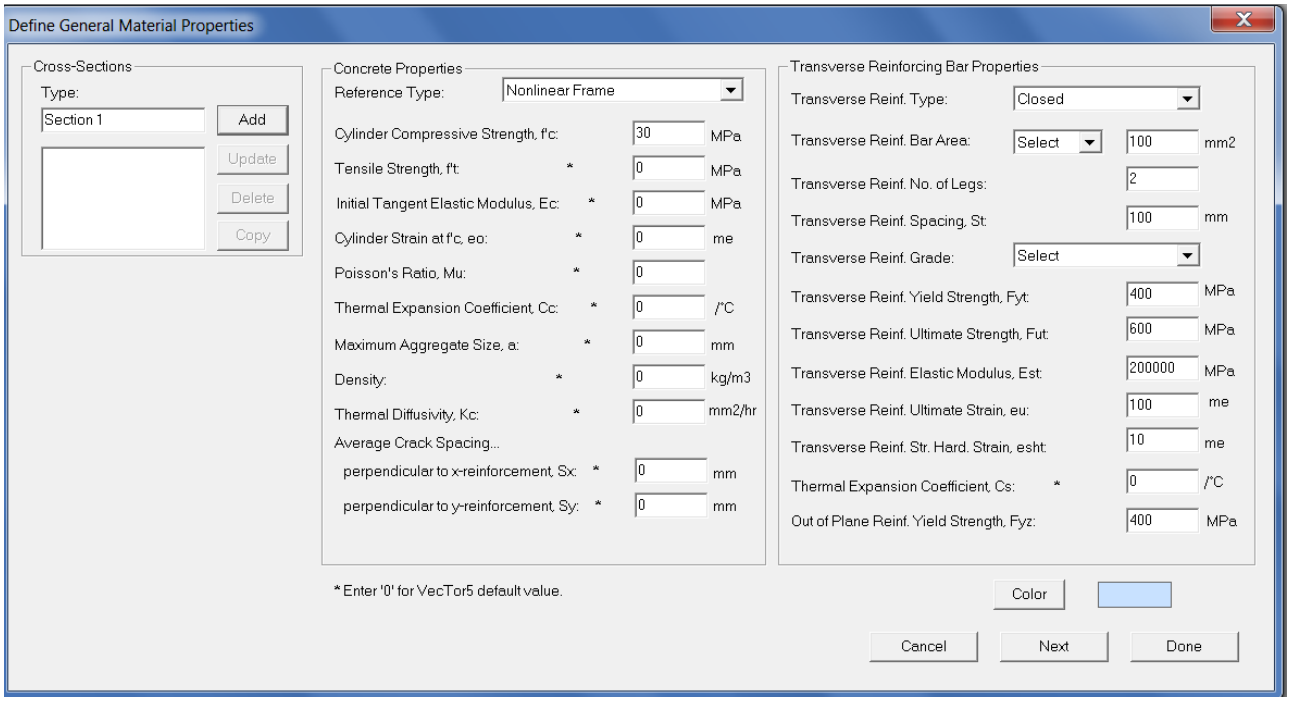


The next tab on this screen is the 'Material Behavior Models' tab. There are several options, which should typically left as default, except the concrete pre-peak response model. Typically, Hognestad Parabola and Popovics (NSC) are used for normal-strength concrete (f'c < 50 MPa). When creating a structure with high-strength concrete (f'c > 50 MPa), the Popovics (HSC) or Hoshikuma model should be selected. For more information about these options, refer to pages 10 and 11 of the *User's Manual of VecTor5*.

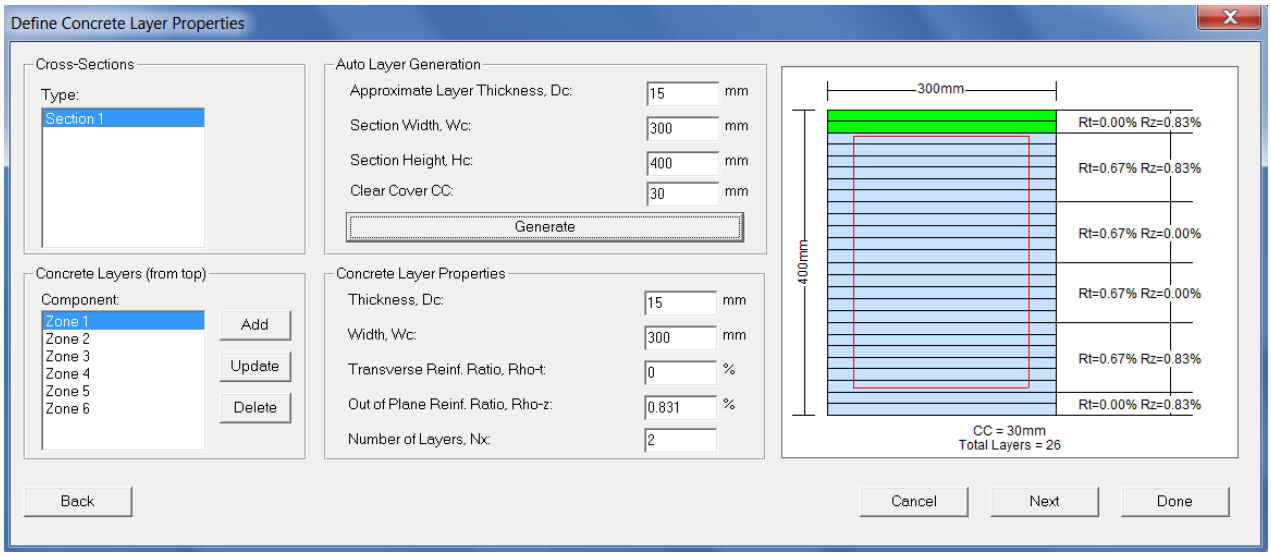


2.2 Define Material Properties & Cross-Sections

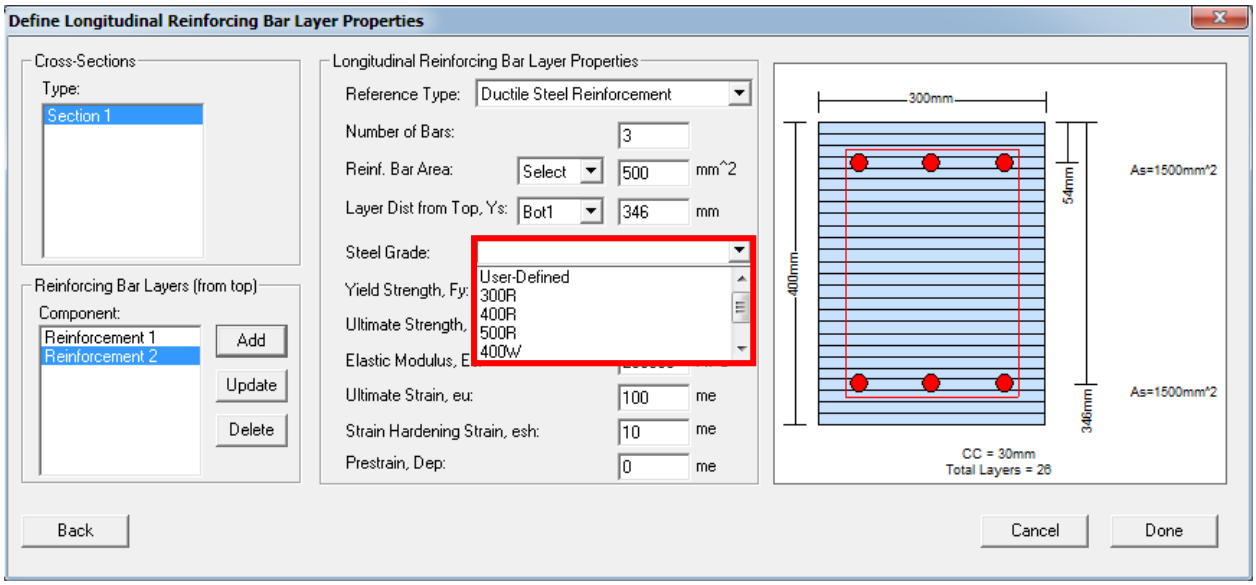
The next menu is to *Define Material Properties*. In this menu, the user will create the cross-sections for the members that will be meshed throughout the structure. The concrete compressive strength, tensile strength, cylinder strain, among other values can be input. Note that the values required for input depends on the material model being selected (Hognestad Parabola, Popovics, etc. – refer to pages 10 and 11 of the *User's Manual of VecTor5*). If the values with an asterisks are left as zero, they will be calculated by the program VecTor5. In addition, the shear reinforcement properties are input in this menu on the right side of the dialog box. If the material properties are known (i.e., from an experimental testing), input them fully into this menu. If the properties are unknown, a steel grade can be selected via the dropdown menu. 300R refers to steel with a yield strength of 300 MPa or 43.5 ksi; 400R refers to 400 MPa or Grade 60 steel (58 ksi); 500R refers to 500 MPa or Grade 70 steel (72.5 ksi); and 400W refers to Grade 60 steel (58 ksi) which is weldable.



Upon clicking "Next", the following menu appears. This menu is used to generate the concrete layers for the cross-section, which is based on the properties of the stirrups that are specified and the dimensions of the cross-section. Four input fields above the generate button should be filled out. It is recommended to specify a layer thickness to provide approximately 30 concrete layers. The values for Rho-t and Rho-z are automatically calculated upon clicking the "Generate" button.



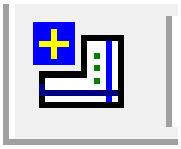
Clicking "Next" will display the following dialog box. In this menu, the longitudinal reinforcement is defined. To create the reinforcement, enter the properties and the Ys distance from the top of the cross section, and simply click "Add". For the top and bottom reinforcement adjacent to the stirrups, Top1 and Bot1 options can be used. For all other layers, calculate and input the Ys distances manually. Once the reinforcement has been created for each section, click "Done" to exit the menu. If the properties are unknown, a steel grade can be selected, as explained above.

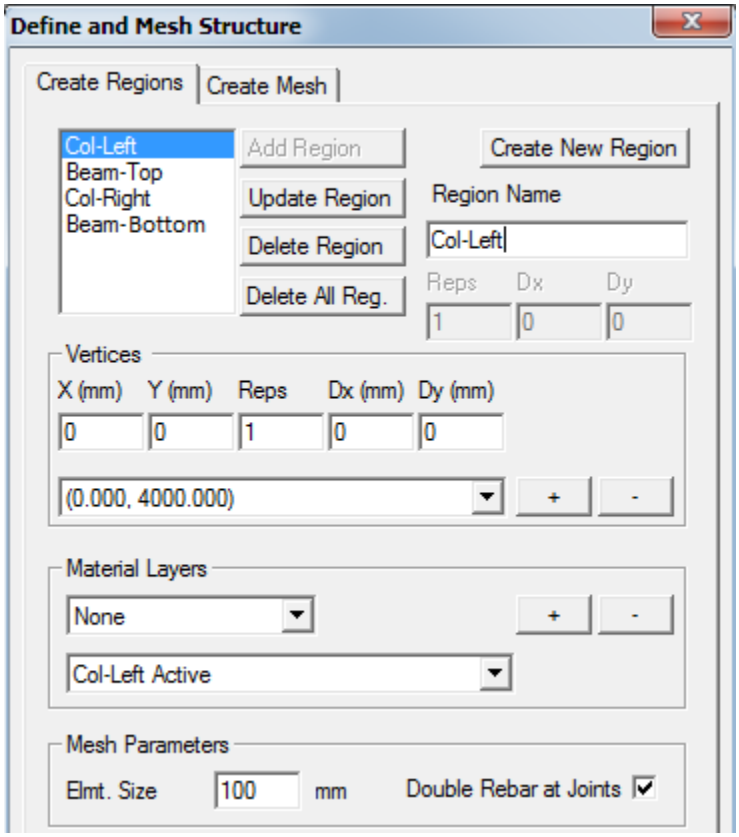


2.3 Define and Mesh Structure

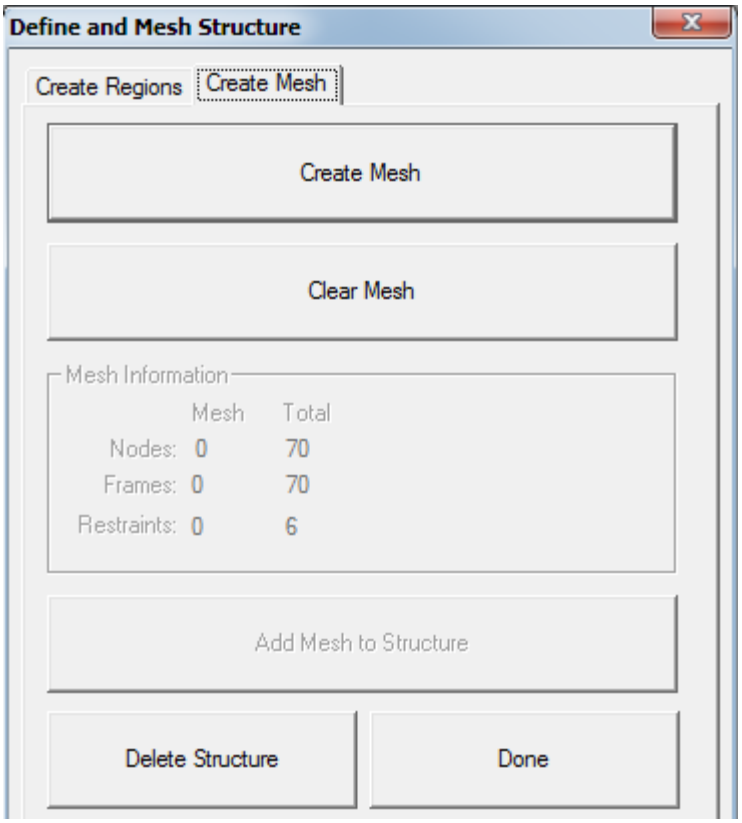
Once the materials and cross-sections have been added, the next step is to open the *Define and Mesh Structure* menu, shown below. This dialog box allows the user to enter the coordinates of the structure to determine which member types (i.e., cross-sections) span between each set of coordinates. FormWorks Plus will mesh the structure and create the members automatically. To begin, click "Create New Region" and add the X and Y-coordinates for which the member type will be assigned. Then, in the Material Layers area, select the cross-section that will span between these coordinates.

Click “Add Region” to incorporate this region into the structure. Regions can be added as needed to achieve the desired structure. One region for each beam or each column is typically required as minimum. If the cross section or reinforcement details change within a beam or column, more regions are needed. If a point load is to be applied, a node should be defined at the load application point. The menu shown below represents an example of how these regions might be defined when creating a model. As previously mentioned, be sure to check “Double Rebar at Joints” to use semi-rigid end offsets. Note that the member length is automatically calculated to be approximately half of the cross section. Optionally, if an exact member length is desired, the user can input the exact number as Elmt. Size. The optimum member length is between 40% and 60% of the cross-sectional height.



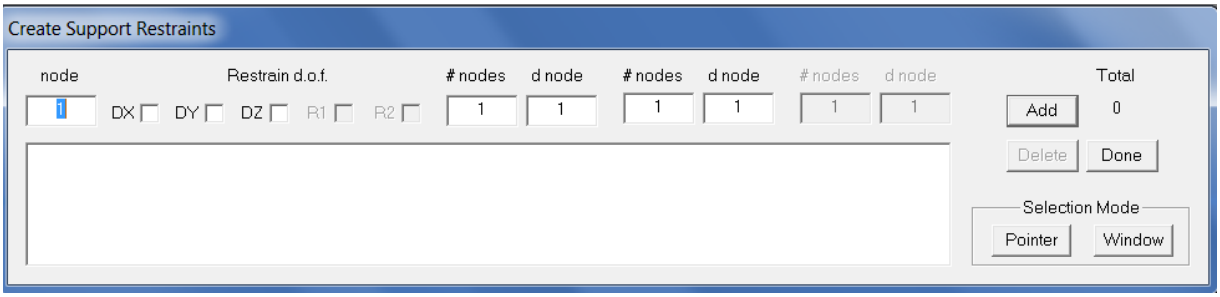


Once the regions have been added, move to the ‘Create Mesh’ tab. Once there, click “Create Mesh” and “Add Mesh to Structure” to allow FormWorks Plus to form the structure with nodes and members. The example shown below represents a structure with 70 nodes and 70 members, with 6 restraints. Then, click “Done” to exit the *Define and Mesh Structure* menu.



2.4 Create Support Restraints

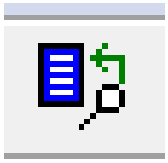
Once the structure has been created, the *Create Support Restraints* menu must be opened. In this menu, the user will define the supports for each node based on the degrees of freedom to be restrained in each direction. To add a support, select the node number, degrees of freedom, and click "Add". Note that the Pointer and Window selection modes are also available.

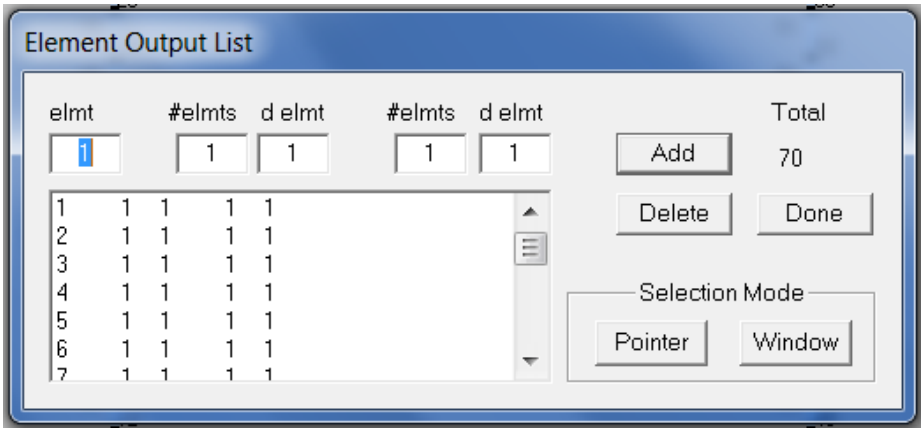


2.5 Element Output List

The *Element Output List* is important for viewing the analysis results in the Janus postprocessor. All elements should typically be selected for the detailed output. To create the output list, click on the

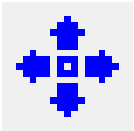
Window option and select the entire structure. The following menu shows an example list from a structure that was modeled using FormWorks Plus.

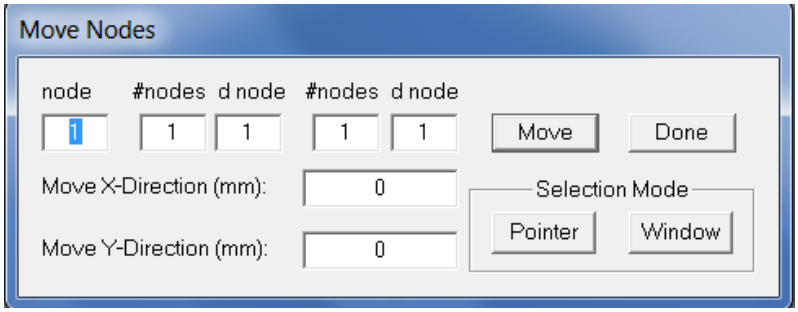




2.6 Move Nodes

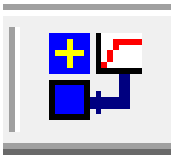
This menu is optional and can be used to slightly adjust the node coordinates. To move a node, enter the node number and the distance in the X and Y-directions. Note that you cannot add a node using this process. The Pointer and Window selection modes are also available.

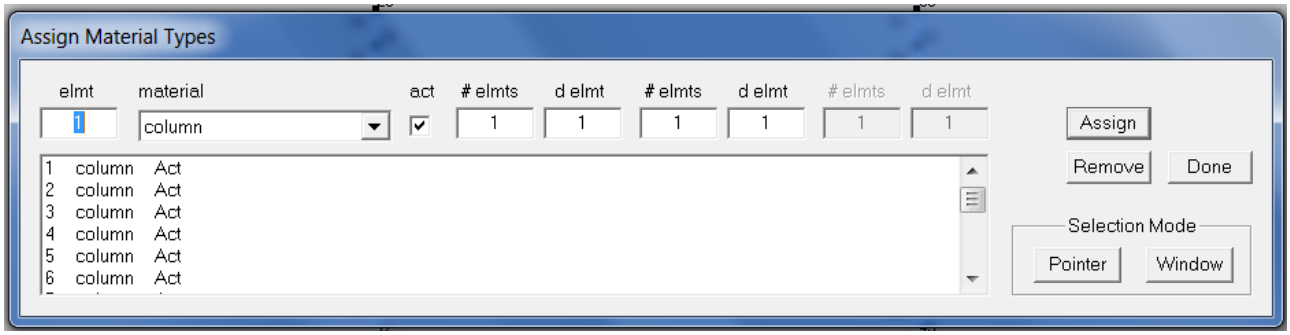




2.7 Reassign Material Types

When meshing a structure, the materials are assigned to the elements as the regions are created. The *Assign Material Types* menu is useful for changing the material type of an element(s) to correct a mistake or to create new regions after the structure is meshed. Note that the elements can be selected via the Pointer and Window selection modes.



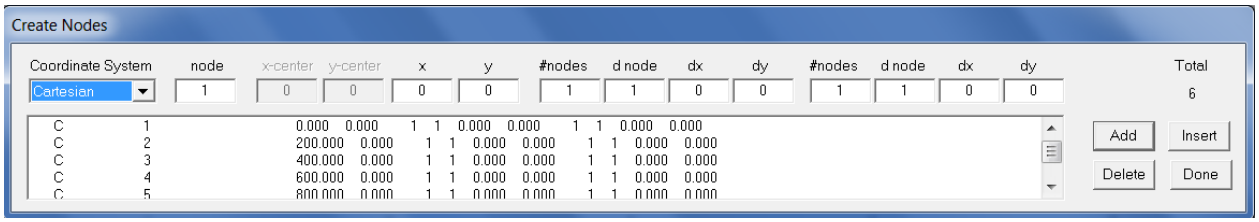


2.8 Optional Advanced Features

FormWorks Plus offers various features that are not necessary for a beginner, but may be valuable for the experienced users under certain situations. Please note that these options cannot be used with the auto-meshing feature, which is the recommended procedure for typical structures.

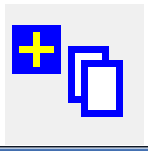
2.8.1 Create Nodes

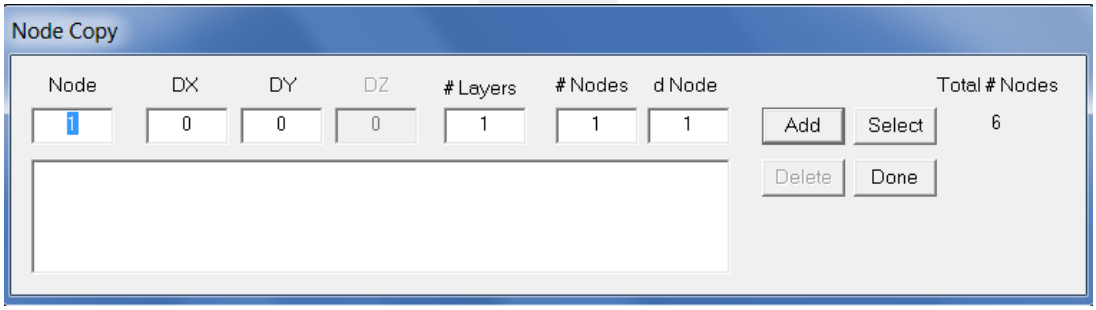
If desired, the user can view and create nodes using the *Create Nodes* menu shown below. To create a node, enter the coordinates of the node and click "Add". This menu is not necessary, but is optional if more nodes are desired. The menu below shows an example of nodes from a structure that was modeled using FormWorks Plus.



2.8.2 Node Copy

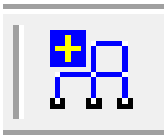
Another menu which is optional, and typically not required, is the *Node Copy* menu. The user can copy nodes that have been created within the structure. To copy a node, enter the node number and the distance in the X and Y-directions.

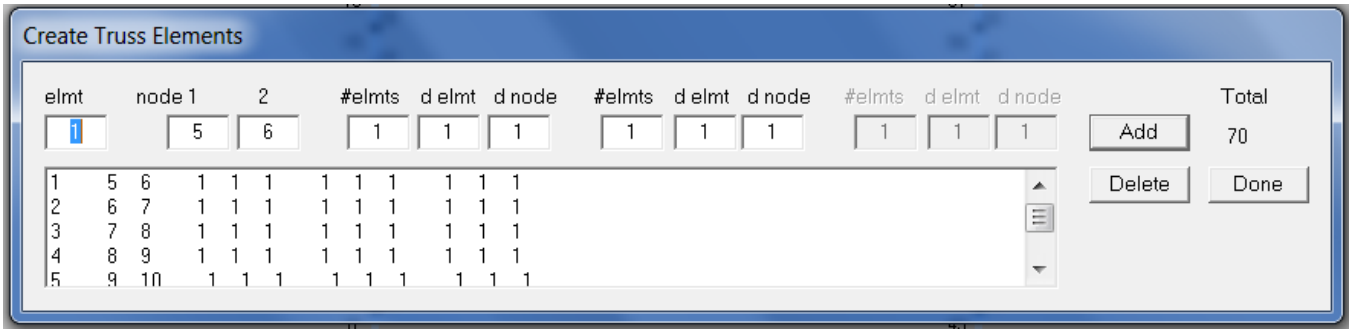




2.8.3 Create Frame Elements

When auto-meshing the structure, the frame elements are automatically created. This menu is for manually creating each element without using the auto-mesh feature. Shown below is the icon and a screenshot of the menu. To add an element, type the element number and choose the nodes that it must span between. The menu shown below is from a structure that was modeled using FormWorks Plus.





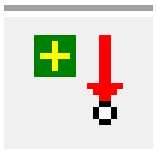
3. APPLYING LOADS

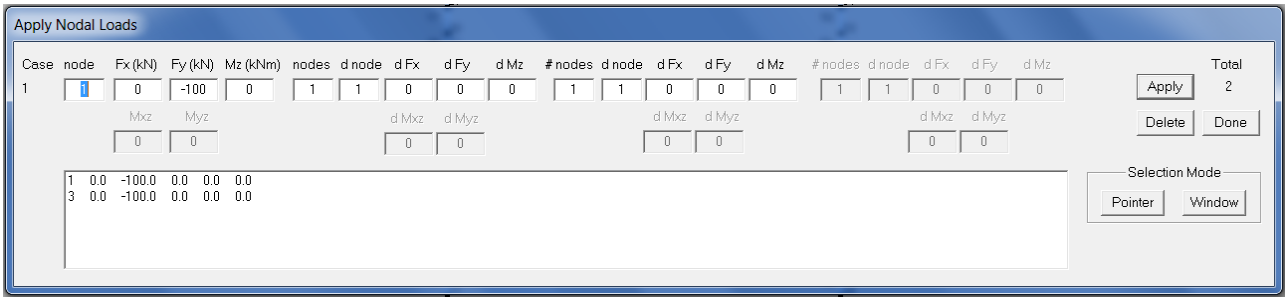
Now that the structure has been created, it is time to apply various loads so that an analysis can be performed. Each type of available loading is discussed. Note that each load case can have multiple loadings associated with it. Be sure to input the correct loading factors and increments in the Job file, so that the structure is loaded properly.

3.1 Apply Nodal Loads

The first available load type is applying a load to the nodes of a structure. To apply a nodal load, click on the following icon to display the corresponding menu. Then, enter the desired node and the loading

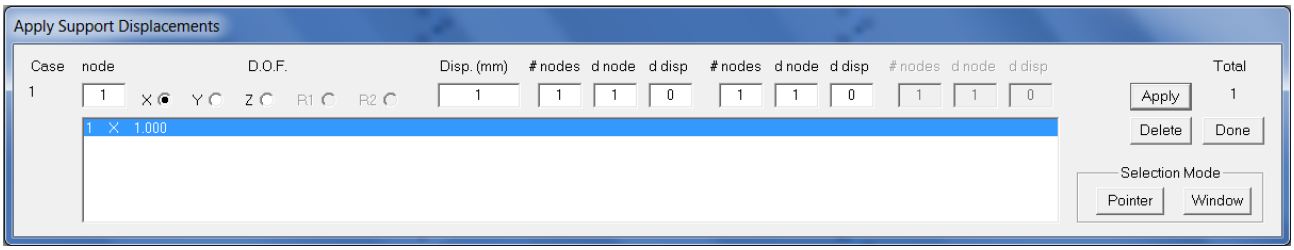
to be applied in the corresponding direction. Positive moments are in the counterclockwise direction. Nodes can also be selected via the Pointer and Window selection modes.





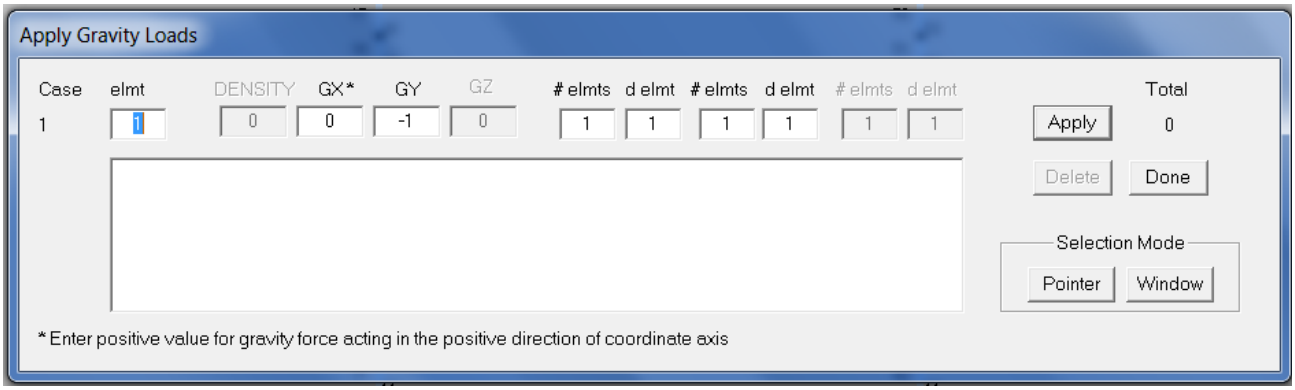
3.2 Apply Support Displacements

Another load case option is to apply a support displacement. This is useful when performing a displacement-controlled analysis. Enter the desired node, and select the direction for the displacement to be applied by clicking the direction under the D.O.F. Enter the displacement amount, which may be 1 millimeter for a displacement-controlled analysis, into the appropriate box. This value will be controlled through the Job file. Note that the nodes can also be selected via the Pointer and Window selection modes.



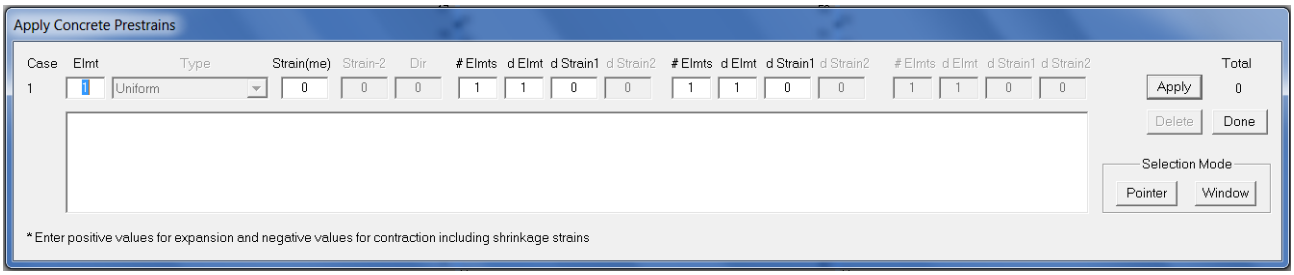
3.3 Apply Gravity Loads

The next loading option is to apply gravity loads throughout the structure. This loading refers to the *self-weight* of the structure. To apply a gravity load, choose the icon shown below. In the menu, enter the factor of gravity load, which is typically -1 in the y-direction. Then, choose the elements to which the load is to be applied, which can be done via the Pointer and Window selection modes. Typically, all elements should be selected. Please note that for a frame structure with slender members, the self-weight will be small and may be neglected.



3.4 Apply Concrete Prestrains

Another loading option is to apply prestrain, or shrinkage strain, to the concrete. To apply the prestrain, choose the following icon to open to menu. Then, enter the elements to which the strain is to be applied, and the value for strain. A typical value of -0.2 millistrain (me) represents a small shrinkage strain. Note that the elements can also be selected via the Pointer and Window selection modes.

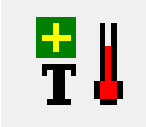


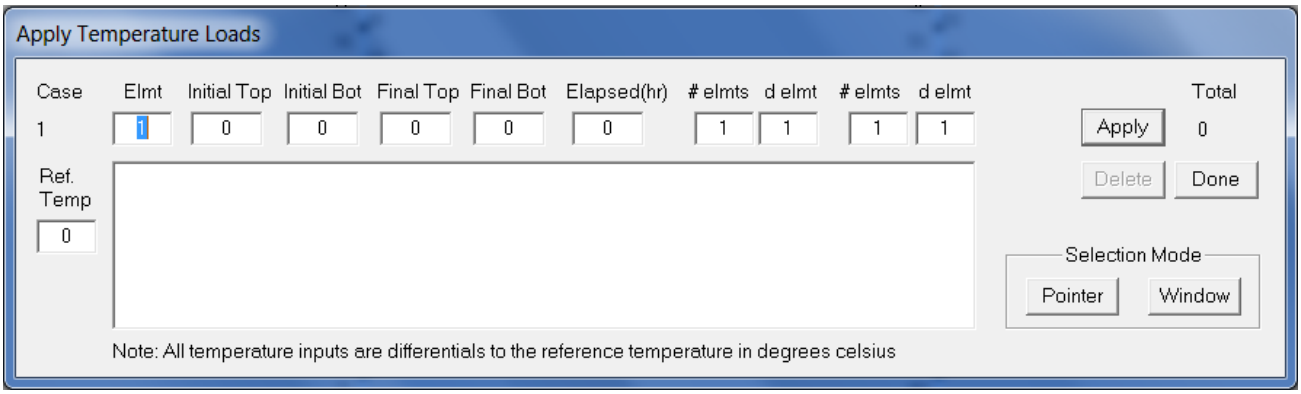
3.5 Apply Impulse Forces

The user can also choose to apply impulse forces to the structure. These forces are used to perform a dynamic impulse analysis. In this menu, enter the desired nodes, degrees of freedom, as well as the impulse force F (kN) and the corresponding time T (s) to be applied. Note that the nodes can also be selected via the Pointer and Window selection modes.

3.6 Apply Temperature Loads

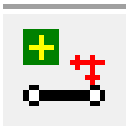
If loading due to temperature changes or fire is present, choose the following icon to show the menu. Enter the desired elements, as well as the temperature differential and the elapsed time. The reference temperature refers to room temperature and the elapsed time is the exposure time. Note that the elements can also be selected via the Pointer and Window selection modes.

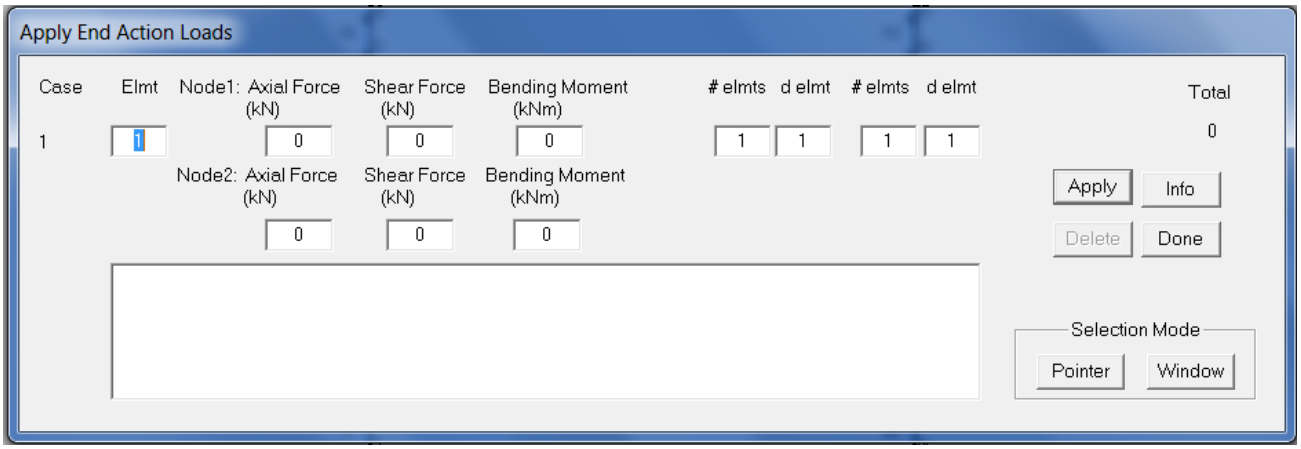


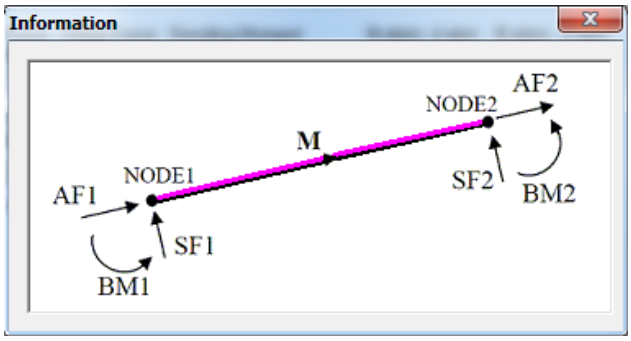


3.7 Apply End Action Loads

Another form of loading is to apply end action loads to the structure. This can be done by selecting the following icon to show the menu. This loading is used less frequently. Enter the desired element, and the axial, shear, and bending moment for each node of the element. The info button gives more details on this load. Note that the elements can also be selected via the Pointer and Window selection modes.

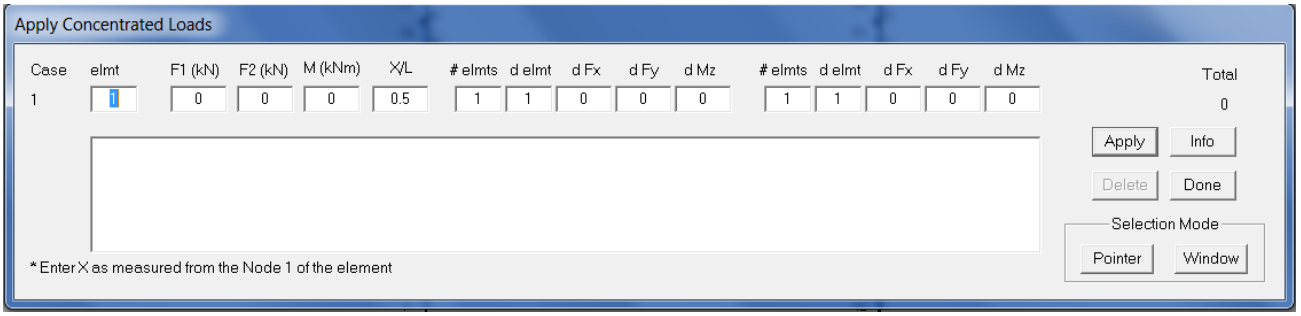


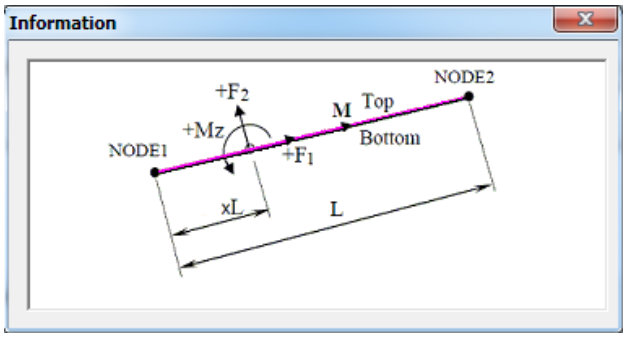




3.8 Apply Concentrated Loads

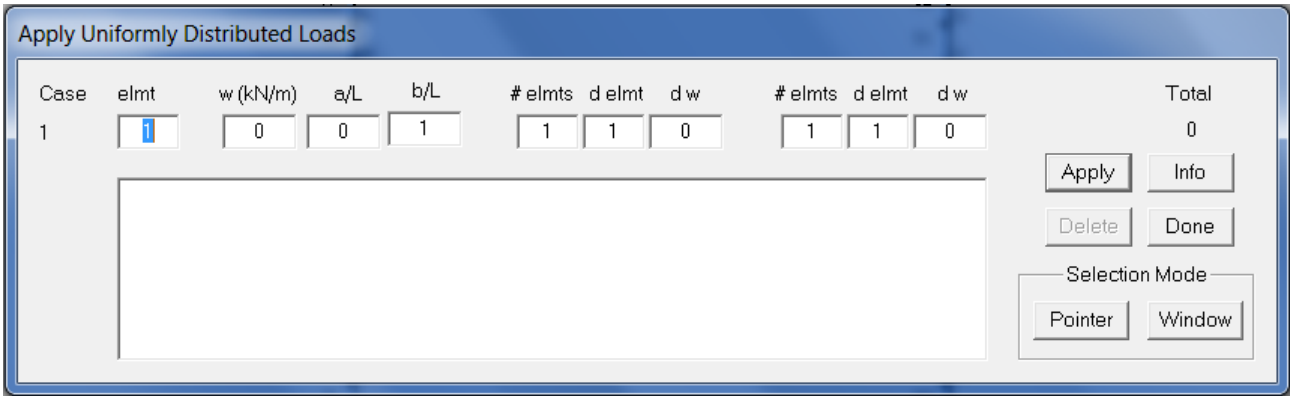
The next type of loading is a concentrated load. This option is useful when a point load is applied in between the nodes. Upon clicking the following icon, the menu appears. Enter the desired elements, and the forces, moments and X/L ratio. The information button gives more details on this load. Note that the elements can also be selected via the Pointer and Window selection modes.

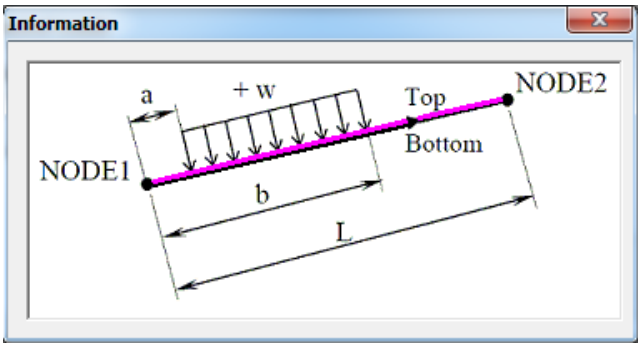




3.9 Apply Uniformly Distributed Loads

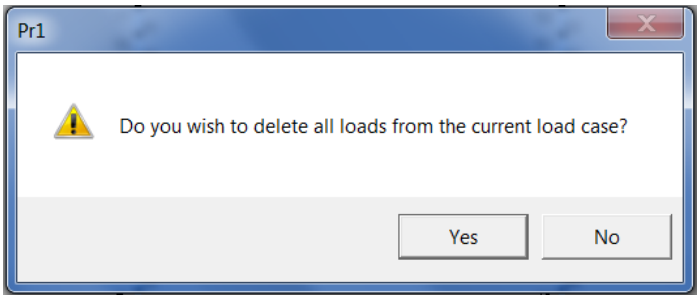
The user can also apply uniformly distributed loads to the structure. This load is useful when modeling the floor loading. To do this, click the following icon to show the menu. Enter the desired elements for the load to be applied, as well as the value for the load and the a/L and b/L ratios. The information button gives more details on this load. Note that the elements can also be selected via the Pointer and Window selection modes.





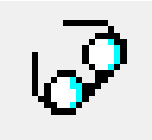
3.10 Clear Load Case

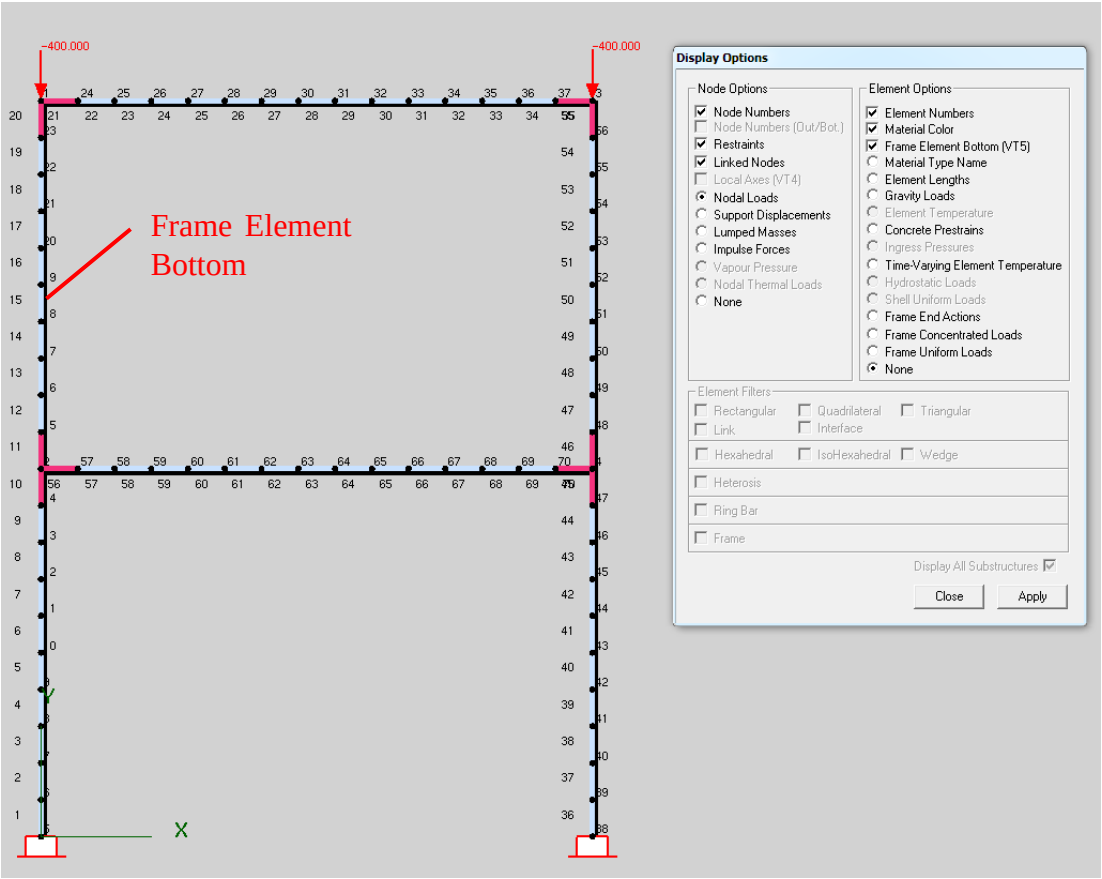
The final icon is located at the far right end of the taskbar. This icon allows the user to clear the current load case that is selected. This may be useful if the user wishes to delete or replace an entire load case. Upon clicking this icon, the following message appears on the screen. Choosing “Yes” will clear the load case from all members and nodes.

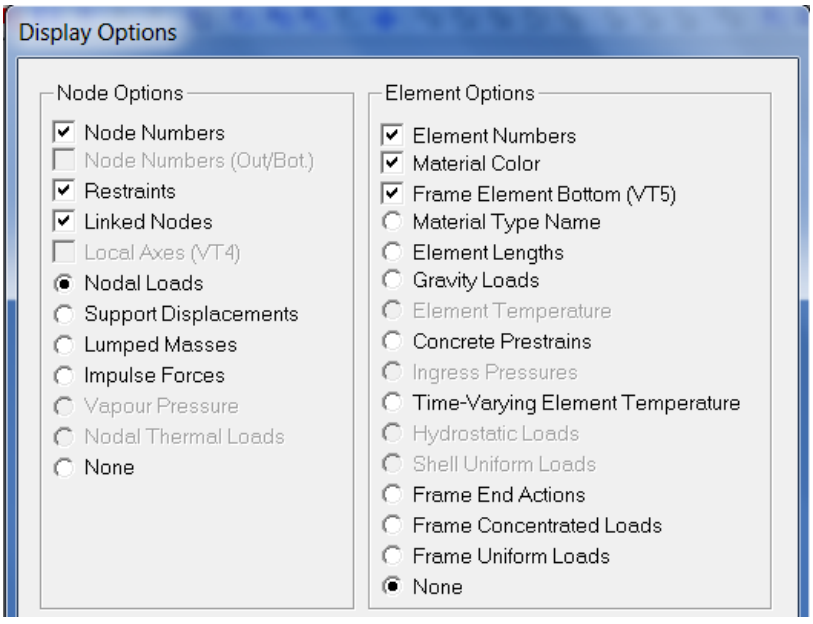


4. DISPLAY OPTIONS

The *Display Options* menu is useful for viewing the structure in FormWorks Plus. There are options to display node and element information. One of the most useful features of this menu is the ability to show the bottom of the element. This way, the user can ensure that each element is oriented properly throughout the structure. To do this, be sure to check the “Frame Element Bottom (VT5)” checkbox. The bottom of each element is indicated with a black line as shown in the image below.

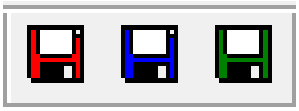



-400.000
-400.000
Frame Element
Bottom
Y
X
Display Options
Node Options
Node Numbers
Node Numbers (Out/Bot.)
Restraints
Linked Nodes
Local Axes (VT4)
Nodal Loads
Support Displacements
Lumped Masses
Impulse Forces
Vapour Pressure
Nodal Thermal Loads
None
Element Options
Element Numbers
Material Color
Frame Element Bottom (VT5)
Material Type Name
Element Lengths
Gravity Loads
Element Temperature
Concrete Prestrains
Ingress Pressures
Time-Varying Element Temperature
Hydrostatic Loads
Shell Uniform Loads
Frame End Actions
Frame Concentrated Loads
Frame Uniform Loads
None
Element Filters
Rectangular
Quadrilateral
Triangular
Link
Interface
Hexahedral
IsoHexahedral
Wedge
Heterosis
Ring Bar
Frame
Display All Substructures
Close
Apply


Display Options
Node Options
Node Numbers
Node Numbers (Out/Bot.)
Restraints
Linked Nodes
Local Axes (VT4)
Nodal Loads
Support Displacements
Lumped Masses
Impulse Forces
Vapour Pressure
Nodal Thermal Loads
None
Element Options
Element Numbers
Material Color
Frame Element Bottom (VT5)
Material Type Name
Element Lengths
Gravity Loads
Element Temperature
Concrete Prestrains
Ingress Pressures
Time-Varying Element Temperature
Hydrostatic Loads
Shell Uniform Loads
Frame End Actions
Frame Concentrated Loads
Frame Uniform Loads
None

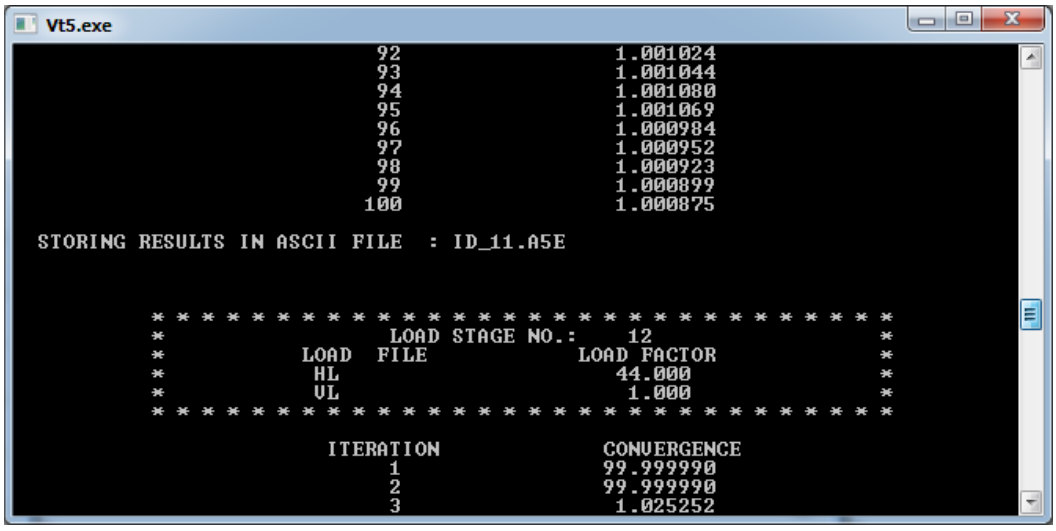
5. PERFORMING AN ANALYSIS

Once the structure and loading have been created, you can perform a VecTor5 analysis. For this, you need to save the input text files created by FormWorks Plus into the folder which contains VT5.exe program. To save the individual files, click on the following save icons. The red icon saves the Job file, the blue icon saves the Structure file, and the green icon saves the Load file.



Now that the files have been saved, the VecTor5 analysis program can be activated using the following icon. This icon is located adjacent to the save icons discussed above. Alternatively, the analysis may be initiated by going into the folder and double clicking on VT5.exe file.

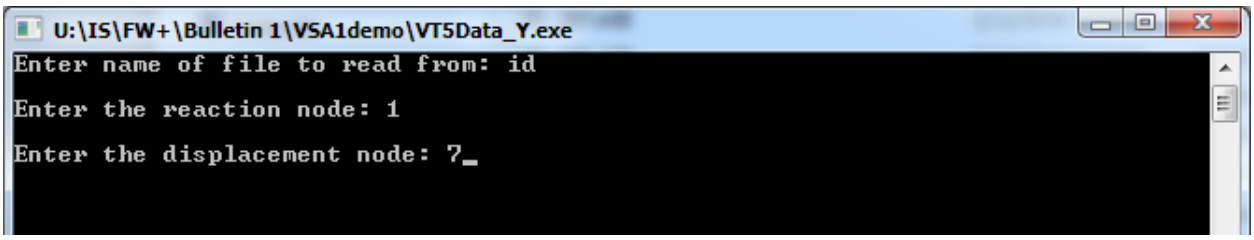
Upon clicking on the VT icon, the following screen appears, showing that the processor VecTor5 is performing the calculations and creating an output file for each load stage. These files are readily available for review in the folder where all files are saved, which is shown below. These files can be opened and viewed using Notepad or other text editors. Expanded data (.S5E) and expanded load files (.L5E) are particularly important to check for the input and modeling mistakes. They list all input in a tabular format for visual inspection. Material properties calculated by VecTor5 should also be checked for accuracy using the .S5E file. For example, check the Ec and e0 values to ensure that they match the values that you wish to use. For more information about the VecTor5 processor and the files it produces, please refer to the *User's Manual of VecTor5*.



VT5	4/5/2016 6:25 PM	Application	1,627 KB
VT5.AUX	4/21/2016 8:23 PM	AUX File	2 KB
HL.L5E	4/21/2016 8:23 PM	L5E File	2 KB
VL.L5E	4/21/2016 8:23 PM	L5E File	2 KB
HL.l5r	4/21/2016 8:23 PM	L5R File	4 KB
VL.l5r	4/21/2016 8:23 PM	L5R File	4 KB
Struct.S5E	4/21/2016 8:23 PM	S5E File	27 KB
Struct.s5r	4/21/2016 8:23 PM	S5R File	20 KB
VecTor	4/21/2016 8:23 PM	Task Scheduler Tas...	3 KB
ID_01.A5E	4/21/2016 8:23 PM	A5E File	935 KB
ID_02.A5E	4/21/2016 8:23 PM	A5E File	935 KB
ID_03.A5E	4/21/2016 8:23 PM	A5E File	936 KB
ID_04.A5E	4/21/2016 8:23 PM	A5E File	936 KB
ID_05.A5E	4/21/2016 8:23 PM	A5E File	936 KB

5.1 Retrieving Output Data

To access the data files that VecTor5 has created, go to the folder where it has saved the files shown above. To obtain the load-deflection data, it is necessary to copy the "VT5Data_X.exe" or "VT5Data_Y.exe" programs into this folder. This folder is also where the Job, Structure, and Load files are saved by the FWP program. Once the data program is copied, open the file to generate the following window. In this window, enter the information as the program prompts. For this example, the reaction node is 1, and the nodal displacement is applied at node 7. In a displacement controlled analysis with a single displacement applied, the same node number should be input for both the reaction and the displacement. This is due to the fact that VecTor5 calculates the reaction for the applied displacement at this node.

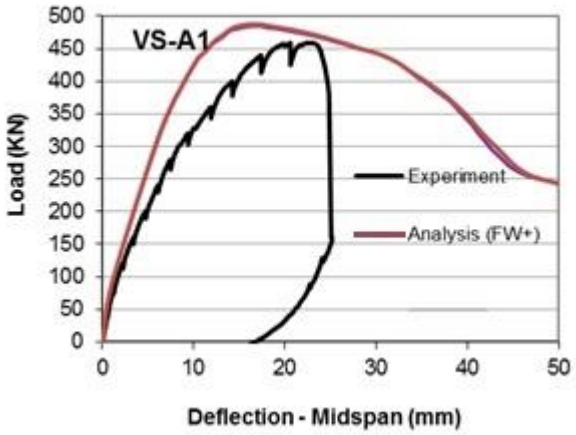


Once the information is entered, click "Enter" to allow the program to read the output data files one by one. Once completed, a text file named "results.dat" will be generated in the same folder. Open this file to view a tabulated list of the load versus deflection data for the beam, respectively. Copy and paste this information into Excel or a graphing program to create charts and graphs to display the data. The Results data file for a beam is shown below, where the nodal displacement is in the left column and the reaction is in the right column.

File Edit Format View Help

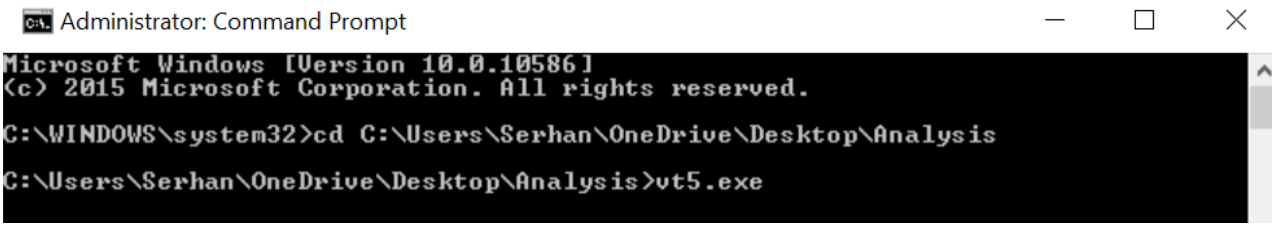
```
Extracted Data for file id

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
```

For this beam, the following graph can be created for the Load versus Deflection data obtained. The load is at node 1, the deflection is at node 7. The experimental data for this beam (VS-A1), was obtained from the journal paper of Vecchio and Shim (2004).



5.2 Troubleshooting

There may be times when the VecTor5 processor displays an error message and will not perform the analysis. In most cases, the black DOS screen will pause and show the error message. If this window closes before displaying the error message, you need to initialize the VT5.exe using the command prompt. For Windows operating systems, right click on the start menu and select Command Prompt. Type CD xxx, where xxx is the full file path of the folder containing all analysis files. You can copy (Ctrl + C) and paste (Ctrl + V) this path from the title bar of Windows File Explorer. Then, type VT5.exe into the DOS window. This will initialize the program and show the error message.

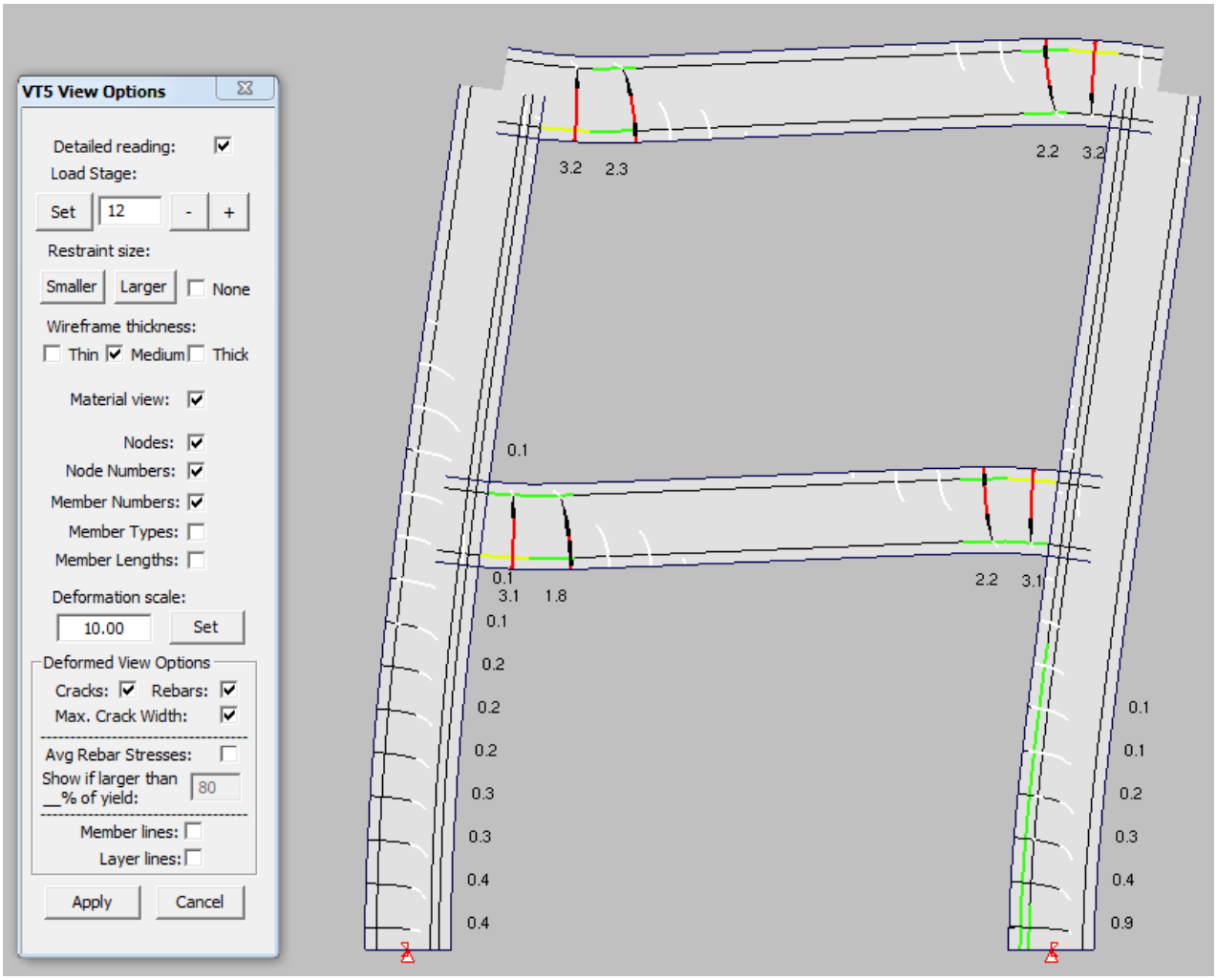


Errors for to the stiffness matrix, such as being not positive definitive, is commonly related to the support conditions. Check the model to ensure that at least three restrains are specified for the structure.

Another common error is the definition of only one load stage in the Job file. In this case, the analysis will terminate after this load stage. Please use more load stages to resolve this problem.

5.3 Visualizing the Analysis Results

To view the results from the VecTor5 analysis, you can use the Janus post-processor, which reads the output text files one by one and displays the results in a graphical manner. First, open the Job file using the top menu of Janus. You will see the *VT5 View Options* menu shown below, which will provide options. You can adjust the view options or change the load stages using the + and – icons. You can also set a particular load stage. The example below shows load stage 12. The crack locations and crack widths can be seen; white lines represent minor cracks of less than 0.1 mm; black is up to 0.1 mm width; bold black up to 1 mm width; and red lines are for cracks larger than 2 mm. For the reinforcement, green is first yielding, yellow is strain hardening, and red lines is for rupture. Refer to the *User's Manual of Janus* and watch-and-learn videos.

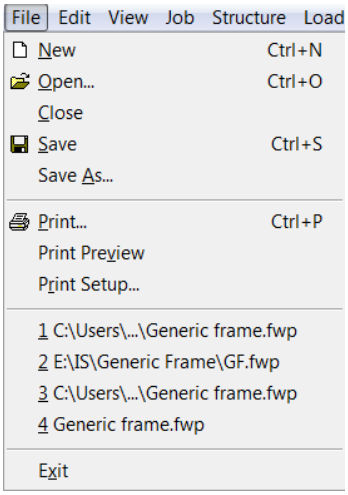


6. TOOLBAR MENUS

An alternate method of creating the structure is using the menus on the top of the screen as discussed below.

6.1 File

The first menu is *File*. This menu allows the user to create a New structure or Open an existing one. Also, the user can Close, Save, Save As, or Print the structure which is currently being created. In this menu, the user can also Exit the program.

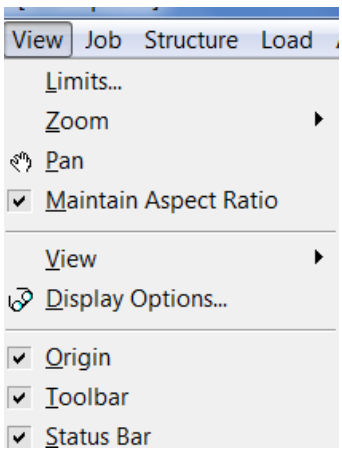


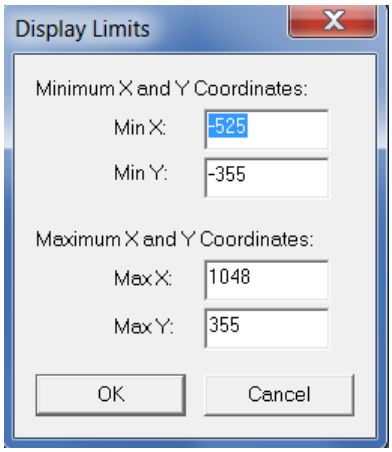
6.2 Edit

This menu is currently inactive.

6.3 View

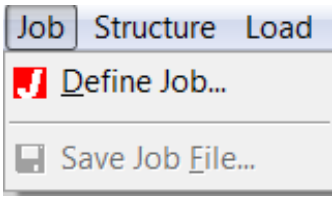
The next tab is the *View* menu, shown below. The user can set the Limits of the display, as shown below. There are also options to Zoom to show the entire structure, a specific window, as well as zoom in and zoom out. There are options to Pan across the screen, and Display Options.





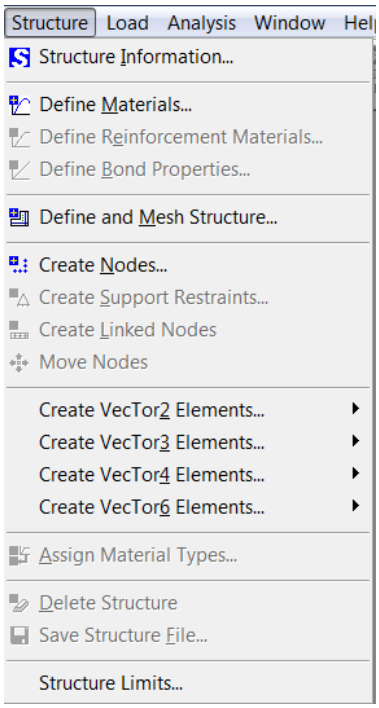
6.4 Job

Next to the *View* menu is the *Job* menu. Here, the user can open the Define Job dialog, and Save the Job File. The Job menu is used to create the *VecTor.JOB* file.



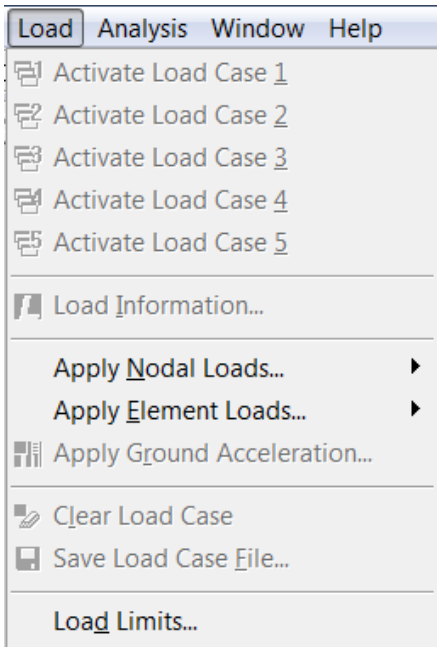
6.5 Structure

The next option is the *Structure* menu. The user can view the Structure Information, Define the Materials, Define and Mesh the structure, create nodes and elements, as well as view the Structure Limits. It is recommended that you use the auto-meshing feature discussed above rather than using this menu.



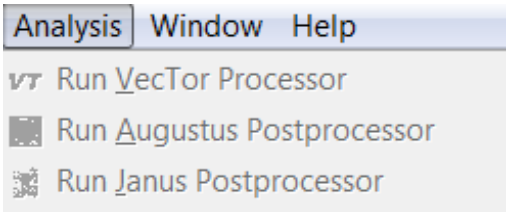
6.6 Load

The following menu refers to the *Load* information. In this menu, the user can activate the Load cases, Apply Nodal Loads, Apply Element Loads, Ground Acceleration, Clear the Load Case, Save the Load Case File (*.L5R*), as well as view the Load Limits of the structure.



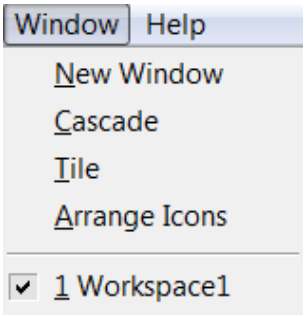
6.7 Analysis

The next menu is for the *Analysis*. In this menu, the user can Run the VecTor Processor and the Janus post-processor. With the Janus postprocessor, the user can view the reactions and failures of the output members. This is a very useful program, which is discussed in its entirety in the *Users Manual for Janus*.



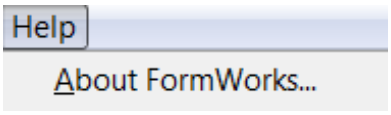
6.8 Window

Next is the *Window* menu. In this menu, the user can create a New Window, Cascade and Tile the windows, as well as Arrange the Icons. This is useful for keeping your work organized.



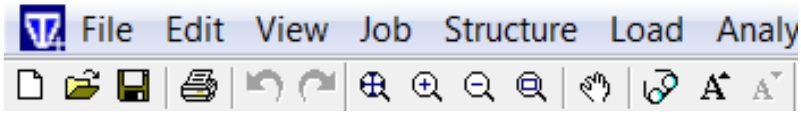
6.9 Help

The final menu in the toolbar is the *Help* menu. Here, the user can view which version of FormWorks Plus is being used to create the structure. The program for this manual is Version 1.0.



6.10 Toolbar Icons

Below the toolbar, there are icons for accessing the premier features of the toolbar menus. The buttons, respectively, are New, Open, Save the Workspace, Print, Zoom All, Zoom In, Zoom Out, Zoom Window, Display Options, Increase Text Size, and Decrease Text Size. It is very important to frequently save the workspace file while creating the structure to avoid losing your work.



7. CONCLUSION

This manual allows the FormWorks Plus user to become familiar with the use the program to prepare text files for an analysis with VecTor5. Using FormWorks Plus to prepare the text files will allow VecTor5 analyses to be performed on structures of any complexity. To better understand the VecTor5 program and the process behind the FormWorks Plus graphical interface, it is highly recommended to read the *User's Manual of VecTor5* and the *User's Manual of Janus for VecTor5*, both of which can be downloaded here. Watch and Learn videos can be found here. These videos show how to use the programs to model various structures.