# 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.	Int	roduction4
2.	Cre	eate A Structural Model 4
	2.1	Define Job
1	2.2	Define Material Properties & Cross-Sections
	2.3	Define and Mesh Structure
	2.4	Create Support Restraints
1	2.5	Element Output List 11
	2.6	Move Nodes
	2.7 R	eassign Material Types
	2.8 O	ptional Advanced Features
	2.8.	1 Create Nodes
	2.8.	2 Node Copy
	2.8.	3 Create Frame Elements
3.	Ap	plying Loads
	3.1	Apply Nodal Loads
	3.2	Apply Support Displacements
	3.3	Apply Gravity Loads
	3.4	Apply Concrete Prestrains
	3.5	Apply Impulse Forces
	3.6	Apply Temperature Loads
	3.7	Apply End Action Loads
	3.8	Apply Concentrated Loads
	3.9	Apply Uniformly Distributed Loads
	3.10	Clear Load Case
4.	Dis	play Options
5.	Per	forming an Analysis
-	5.1	Retrieving Output Data
	5.2	Troubleshooting
-	5.3	Visualizing the Analysis Results
6.	Тос	olbar Menus
(	6.1	File
(	6.2	Edit

7.	Coi	1clusion	. 29
	0.10		
	6.10	Toolbar Icons	. 29
	6.9	Help	. 29
	6.8	Window	. 28
	6.7	Analysis	. 28
	6.6	Load	. 27
	6.5	Structure	. 27
	6.4	Job	. 27
	6.3	View	. 26

# **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 <u>here</u>. Watch and Learn videos can be found <u>here</u>. 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.



ne Job							<u> </u>
ob Control Mode	els Auxiliary Auxi	liary Thermal					
Job Data Job file name: Job title: Date:	VecTor Enter Job Title Enter Date		Structure Data Structure file name Structure title: Structure type:	Enter Struct Enter Structure Tit Plane Frame (2-D)	le V		
Load se	ries ID: ID	Starting load sta	age no.: 1	No. of load	stages: 200		
Activate:	Case 1	Case 2	Case 3	Case 4	Case 5	-	
Load file name:	HL	VL	NULL	NULL	NULL		
Load case title:	Enter load case title	Enter load case title	Enter load case title	Enter load case title	Enter load case title		
Initial factor:	0	1	1	1	1		
Final factor:	500	1	100	100	100		
Inc. factor:	4	0	1	1	1		
Load type:	Monotonic 💌	Monotonic 💌	Monotonic 💌	Monotonic 💌	Monotonic 💌		
Repetitions:	1	1	1	1	1		
Cyclic Inc. factor:	0	0	0	0	0		
Initial Load Stage:	1	1	1	1	1		
Analysis Paramete	rs					1	
	Seed File Name:	NULL	Convergence Criteria	a: Displacements - V	Veighted Average 💌		
Ma	ax. no. of Iterations:	100	Analysis Mode	: Static and Therma	al Analyses 💌		
Dynamic	c Averaging Factor:	0	Results Files	: ASCII Files Only	•		
	Convergence Limit:	1.00001	Modeling Format	: Stand Alone Mode	eling 💌		
						]	
					ОК	Cancel	Apply

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*.

Define Job	
Job Control Models	Auxiliary   Auxiliary Thermal
Concrete Models	
Compression Pre-Peak:	Hognestad (Parabola)
Compression Post-Peak:	Linear Hognestad (Parabola)
Compression Softening:	Popovics (NSC) Popovics (HSC)
	Hoshikuma et al

### 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.



Define General Material Properties			X
Cross-Sections Type: Section 1 Add Update Delete Copy	eference Type: Nonlinear Frame  eference Type: Nonlinear Frame ylinder Compressive Strength, fc: 30 MPa ensile Strength, ft: * 0 MPa vlinder Strain at fc, eo: * 0 me oisson's Ratio, Mu: * 0 hermal Expansion Coefficient, Cc: * 0 /'C leavimum Aggregate Size, a: * 0 mm ensity: * 0 kg/m3 hermal Diffusivity, Kc: * 0 mm2/hr verage Crack Spacing perpendicular to x-reinforcement, Sx: * 0 mm Enter '0' for VecTor5 default value.	Transverse Reinforcing Bar Properties         Transverse Reinf. Type:       Closed         Transverse Reinf. Bar Area:       Select         Transverse Reinf. Bar Area:       Select         Transverse Reinf. No. of Legs:       2         Transverse Reinf. Spacing. St.       100         Transverse Reinf. Grade:       Select         Transverse Reinf. Grade:       Select         Transverse Reinf. Grade:       Select         Transverse Reinf. Ultimate Strength, Fyt:       400         Transverse Reinf. Ultimate Strength, Fut:       600         Transverse Reinf. Ultimate Strain, eu:       100         Transverse Reinf. Str. Hard. Strain, esht:       10         Transverse Reinf. Str. Hard. Strain, esht:       10         Transverse Reinf. Yield Strength, Fyz:       400         Out of Plane Reinf. Yield Strength, Fyz:       400	mm2 mm MPa MPa me me MPa

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.

- Cross-Sections						
Туре:	Approximate Layer Thickness, Dc:	15	mm	⊢	300mm	
Section 1	Section Width, Wc:	300	mm			Rt=0.00% Rz=0.83%
	Section Height, Hc:	400	mm			Rt=0.67% Rz=0.83%
	Clear Cover CC:	30	mm			
	Generate				 	Rt=0.67% Rz=0.00%
- Concrete Layers (from top)	Concrete Layer Properties			400m		
Component:	Thickness, Dc:	15	mm			Rt=0.67% Rz=0.00%
Zone 1 Add	Width, Wc:	300	mm			
Zone 3 Zone 4 Update	Transverse Reinf. Ratio, Rho-t:	0	%			Rt=0.67% Rz=0.83%
Zone 5 Zone 6 Delete	Out of Plane Reinf. Ratio, Rho-z:	0.831	%			Rt=0.00% Rz=0.83%
	Number of Layers, Nx:	2			CC = 30mn Total Layers =	n : 26
Back					Cancel N	ext Done

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.

Define Longitudinal Reinforcing Bar La	er Properties	X
Cross-Sections Type: Section 1	Longitudinal Reinforcing Bar Layer Properties Reference Type: Ductile Steel Reinforcement Number of Bars: 3 Reinf. Bar Area: Select 500 mm <sup>2</sup> Layer Dist from Top, Ys: Bot1 346 mm Steel Grade:	
Reinforcing Bar Layers (from top) Component: Reinforcement 1 Reinforcement 2 Update Delete	Yield Strength, Fy:       User-Defined         300R       400R         400R       500R         Elastic Modulus, E       400W         Ultimate Strength, 500R       500R         Ultimate Strain, eu:       100         Strain Hardening Strain, esh:       10         Prestrain, Dep:       0	:
Back	Cancel Done	

# 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.

	:		

fine and Mesh Structure					
Create Regions Cr	eate Mesh				
Col-Left	Add Region	Create New Region			
Col-Right	Update Region	Region Name			
Beam-Bottom	Delete Region	Col-Left			
	Delete All Reg.	Reps Dx Dy			
Vertices		ji ju ju			
X (mm) Y (mm)	Reps Dx (mm)	Dy (mm)			
0	1 0	0			
(0.000, 4000.000	)	▼ + -			
Material Lawren					
Material Layers					
Nene	-				
None	•	+ -			
None Col-Left Active	•	+ -			
Col-Left Active		+ -			

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.

efine and Mesh Struc	fine and Mesh Structure							
Create Regions Creat	e Mesh							
	Create Mesh							
	Clear	r Mesh						
Mesh Information —								
Mesh	Total							
Nodes: 0	70							
Frames: 0	70							
Restraints: 0	6							
,	Add Mesh	to Structure						
Delete Structu	re	Done						

# **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.



Create Suppo	ort Restraints			
node	Restrain d.o.f.	#nodes dinode	# nodes     d node     # nodes     d node       1     1     1     1	Total Add 0
				Selection Mode Pointer Window

# **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.



Element Output List		
elmt #elmts d elmt	#elmts d elmt	Total Add 70
$ \begin{array}{cccccccccccccccccccccccccccccccccccc$	▲ Ⅲ	Delete Done
4 1 1 1 1	Г	Selection Mode
$\begin{bmatrix} 5 & 1 & 1 & 1 & 1 \\ 6 & 1 & 1 & 1 & 1 \\ 7 & 1 & 1 & 1 & 1 \end{bmatrix}$	-	Pointer Window

#### 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.



Assign Material Types		
elmt material	act #elmts delmt #elmts delmt	# elmts d elmt
1 column Act 2 column Act 3 column Act		Remove Done
4 column Act 5 column Act		Pointer Window
		· · · · · · · · · · · · · · · · · · ·

#### **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.

_	
	- <b>4</b> -
	- <b>1</b> -
	. T.
- <b>-</b>	**

Create Nodes		
Coordinate System node Cartesian 💌 1	x-center         y-center         x         y         #nodes         d node         dx         dy         #nodes         d node         dx         dy           0         0         0         0         1         1         0         0         1         1         0         0	Total 6
C 1 C 2 C 3 C 4 C 5	0.000 0.000 1 1 0.000 0.000 1 1 0.000 0.000 200.000 0.000 1 1 0.000 0.000 1 1 0.000 0.000 400.000 0.000 1 1 0.000 0.000 1 1 0.000 0.000 600.000 0.000 1 1 0.000 0.000 1 1 0.000 0.000 800.000 0.000 1 1 0.000 0.000 1 1 0.000 0.000 800.000 0.000 1 1 0.000 0.000 1 1 0.000 0.000 800.000 0.000 0 0 0 0 0 0 0 0 0 0 0 0 0	Add Insert Delete Done

# 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.

		•	
Node Copy			
Node [	DX DY DZ 0 0 0	#Layers #Nodes dNode	Total # Nodes Add Select 6 Delete Done

#### **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.



Create	e Truss Elemer	nts				
elm 1	t node 1	2	#elmts d elmt d node	#elmts d elmt d node	#elmts d elmt d node	Total Add 70
1 2 3 4 5	5 6 1 6 7 1 7 8 1 8 9 1 9 10 1	$ \begin{array}{cccccccccccccccccccccccccccccccccccc$	$\begin{array}{cccccccccccccccccccccccccccccccccccc$			Delete Done

# 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.



Apply Nodal Load	ds	1.0		24	
Case node Fx	x (kN) Fy (kN) Mz (kNm) nodes d node 0 -100 0 1 1	d Fx d Fy d Mz # nod	es dinode id Fx id Fy id Mz	#nodes dinode d Fx d Fy d Mz	Total Apply 2
Г	Mxz Myz 0 0	d Mxz d Myz	d Mxz d Myz	d Mxz d Myz	Delete Done
1 0.0 - 3 0.0 -	1000 0.0 0.0 0.0 100.0 0.0 0.0 0.0				Pointer Window

# 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.



ply Support Displacements	A	2	
Case node D.O.F.	Disp. (mm) #nodes dinode didisp #no	nodes dinode didisp ≢nodes dinode didisp 1 1 0 1 1 0	Total Apply 1 Delete Done Selection Mode Pointer Window

# **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.

+	G

Apply	Gravity Loads		1	
Case	e elmt	DENSITY         GX*         GY         GZ         #elmts         d elmt         #elmts         d elmt         #elmts           0         0         -1         0         1         1         1         1         1	d elmt	Total Apply 0
				Delete Done
				Selection Mode Pointer Window
*Ent	, er positive valu	e for gravity force acting in the positive direction of coordinate axis		

# **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.



App	y Coi	ncrete Prest	rains				100	
Ca 1	se [	Elmt	Type rm	Strain(me) Strain-2 Dir	#Elmts dElmt dStrain1 dStrain2	#Elmts d Elmt d Strain1 d Strain2	# Elmts d Elmt d Strain1 d Strain2	Total Apply 0
								Delete Done
								Pointer Window
*E	nter p	oositive value	s for expansic	n and negative values for contractio	n including shrinkage strains			

# 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.



Apply Im	pulse Forces	100			20	
Case 1	node #nodes dinode	#nodes d node	#nodes dnode	d.o.f. X ● Y ○ Z ○	T F	Total Apply 0 Graph Delete Done
						Selection Mode Pointer Window

# **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.



Apply Te	nperature Loads	
Case	Elmt       Initial Top       Initial Bot       Final Top       Final Bot       Elapsed(hr)       # elmts       d elmt         1       0       0       0       0       1       1       1	Total Apply 0
Ref. Temp		Delete Done
		Selection Mode Pointer Window
	Note: All temperature inputs are differentials to the reference temperature in degrees celsius	

# **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.



Apply En	nd Action Loads			
Case	Elmt Node1: Axial Force (kN)	Shear Force Bending Moment (kN) (kNm) 0 0	#elmts delmt #elmts delmt	Total 0
	Node2: Axial Force (kN)	Shear Force Bending Moment (kN) (kNm) 0 0		Apply     Info       Delete     Done
				Selection Mode Pointer Window



# 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.



Apply Co	oncentrated	d Loads			-						
Case 1	elmt	F1 (kN)	F2 (kN)	M (kNm)	X/L 0.5	#elmts delmt	d Fx d Fy	d Mz	#elmts delmt dFx dFy dMz		Total 0
										Apply	Info
										Selection	n Mode
*Enter	Xas measu	ired from th	e Node 1	of the eleme	ent					Pointer	Window



# 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.





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

# 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.

# VT

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.exe	
$\begin{array}{cccccccccccccccccccccccccccccccccccc$	
STORING RESULTS IN ASCII FILE : ID_11.A5E	
* * * * * * * * * * * * * * * * * * *	Ш
ITERATION CONVERGENCE 1 99.999990 2 99.999990 3 1.025252	Ţ

<b>Ⅲ</b> 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.I5r	4/21/2016 8:23 PM	L5R File	4 KB
VL.15r	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
D_01.A5E	4/21/2016 8:23 PM	A5E File	935 KB
D_02.A5E	4/21/2016 8:23 PM	A5E File	935 KB
D_03.A5E	4/21/2016 8:23 PM	A5E File	936 KB
D_04.A5E	4/21/2016 8:23 PM	A5E File	936 KB
D_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.

<u>File E</u> dit F <u>o</u> rmat <u>V</u>	<u>liew H</u> elp
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
0 500000	205 600006

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.

Fil	e Edit	View	Job	Structure	Load	
D	<u>N</u> ew			Ctr	+N	
2	<u>O</u> pen			Ctr	+0	
	<u>C</u> lose					
	<u>S</u> ave			Ctr	I+S	
	Save <u>A</u> s	i				
6	<u>P</u> rint			Ctr	I+P	
	Print Pr	e <u>v</u> iew				
	P <u>r</u> int Se	tup				
	<u>1</u> C:\Use	ers\\G	Generi	c frame.fwp	)	
	<u>2</u> E:\IS\	Generio	: Fram	e\GF.fwp		
	3 C:\Users\\Generic frame.fwp					
	4 Gener	ric fram	ne.fwp	1		
	E <u>x</u> it					

# 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.

	•	DOLLAR DE LA DEL	
View Job Structure Load		Display Limits	
Limits		Minimum $\times$ and $Y$	Coordine
Zoom •		Min X:	-525
《か <u>P</u> an		Min Y:	-355
<ul> <li><u>Maintain Aspect Ratio</u></li> </ul>			
View		Maximum X and Y	'Coordine
		MaxX:	1048
		Max Y:	355
✓ <u>O</u> rigin			1
✓ Toolbar		ОК	Car
Status Par			

#### 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.

Job	Structure	Load
<b>, D</b>	efine Job	
S	ave Job <u>F</u> ile	)

#### 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.

Str	ructure Load Analysis Window	/ Hel
S	Structure Information	
	Define <u>M</u> aterials	
Z	Define Reinforcement Materials	.
Z	Define <u>B</u> ond Properties	
2	Define and Mesh Structure	
•.:	Create <u>N</u> odes	
	Create Support Restraints	
	Create Linked Nodes	
***	Move Nodes	
	Create VecTor2 Elements	•
	Create VecTor3 Elements	•
	Create VecTor4 Elements	•
	Create VecTor <u>6</u> Elements	•
5	<u>A</u> ssign Material Types	
	Delete Structure	
	Save Structure File	
	Structure Limits	

#### 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.

Load	Analysis	Window	Help
립 Ac	tivate Load	l Case <u>1</u>	
문2 Ac	ctivate Load	d Case <u>2</u>	
문 Ad	ctivate Load	l Case <u>3</u>	
립 Ac	ctivate Load	d Case <u>4</u>	
문5 Ac	ctivate Load	l Case <u>5</u>	
M Lo	ad <u>I</u> nforma	ition	
Aŗ	oply <u>N</u> odal	Loads	•
Aı Aı	oply <u>N</u> odal oply <u>E</u> lemer	Loads nt Loads	<b>)</b>
Aş Aş Mi Aş	oply <u>N</u> odal oply <u>E</u> lemer oply G <u>r</u> ound	Loads nt Loads d Accelerat	tion
	oply <u>N</u> odal oply <u>E</u> lemen oply G <u>r</u> ound ear Load Ca	Loads nt Loads d Accelerat ase	► tion
Ar Ar Mi Ar Ar Cl Sa	oply <u>N</u> odal oply <u>E</u> lemen oply G <u>r</u> ound ear Load Ca ave Load Ca	Loads ht Loads d Accelerat ase se <u>F</u> ile	tion

# 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*.

An	alysis	Window	Help
ντ	Run <u>V</u>	ecTor Proc	essor
	Run <u>A</u>	ugustus Po	stprocessor
2 Å	Run <u>J</u> a	anus Postpi	rocessor

#### 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.

Window	Help
New Window	
<u>C</u> ascade	
<u>T</u> ile	
<u>A</u> rrange Icons	
✓ <u>1</u> Wor	kspace1

#### 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.

Help	
<u>A</u> bout FormWorks	

#### 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 VecTor5*, both of which can be downloaded <u>here</u>. Watch and Learn videos can be found <u>here</u>. These videos show how to use the programs to model various structures.