

USER BULLETIN 6

FRAME MODELING WITH VECTOR5 USING PRE-PROCESSING FACILITIES

1. INTRODUCTION

To correctly use VecTor5 with concrete frames, the pre-processing program FormWorks-Plus can be used to generate the input files while the post-processing program Janus can be used to view the results. This pre-processing program allows for the modeling of the frame and also offers the ability to generate the repetitive mathematical data from a small set of inputs. These features help to speed up the input process and help minimize any potential input mistakes. In this document, the [Duong Frame](#) is modeled for monotonic loading conditions using FormWorks-Plus for demonstrative purposes.

Before using this bulletin, please read [User Bulletin 4](#) and [User Manual for FormWorks Plus](#) found in the publications section of [Dr. Guner's website](#). [VecTor5 manual](#) (Guner and Vecchio, 2008) is another useful reference document. These documents will give you the required basis for using both FormWorks-Plus and VecTor5, and help better understand this bulletin.

This document is prepared by Andrew Coffman (third-year undergraduate student at the University of Toledo, OH, USA, as a part of a project supervised by [Dr. Serhan Guner](#).

2. DEFINING THE JOB FILE

The first step is to open the “Define Job” window using the button shown in **Figure 1**.

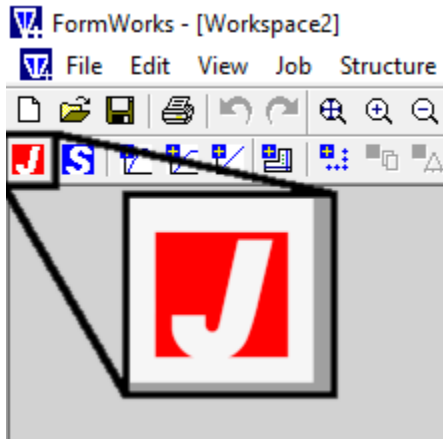


Figure 1 – “Define Job” button.

When the “Define Job” window comes up, ensure that the structure type option is set to “Plane Frame (2-D).” Following the same process as described in User Bulletin 4, configure the Job file with the only difference being that there are two different types of loads being applied to the frame, one being a controlled displacement and the other being a constant force. To use both of these cases, check the box next to Case 2 and fill out its information ensuring that the starting and ending factor are the same as the constant force. A picture the data entered is presented in **Figure 2**.

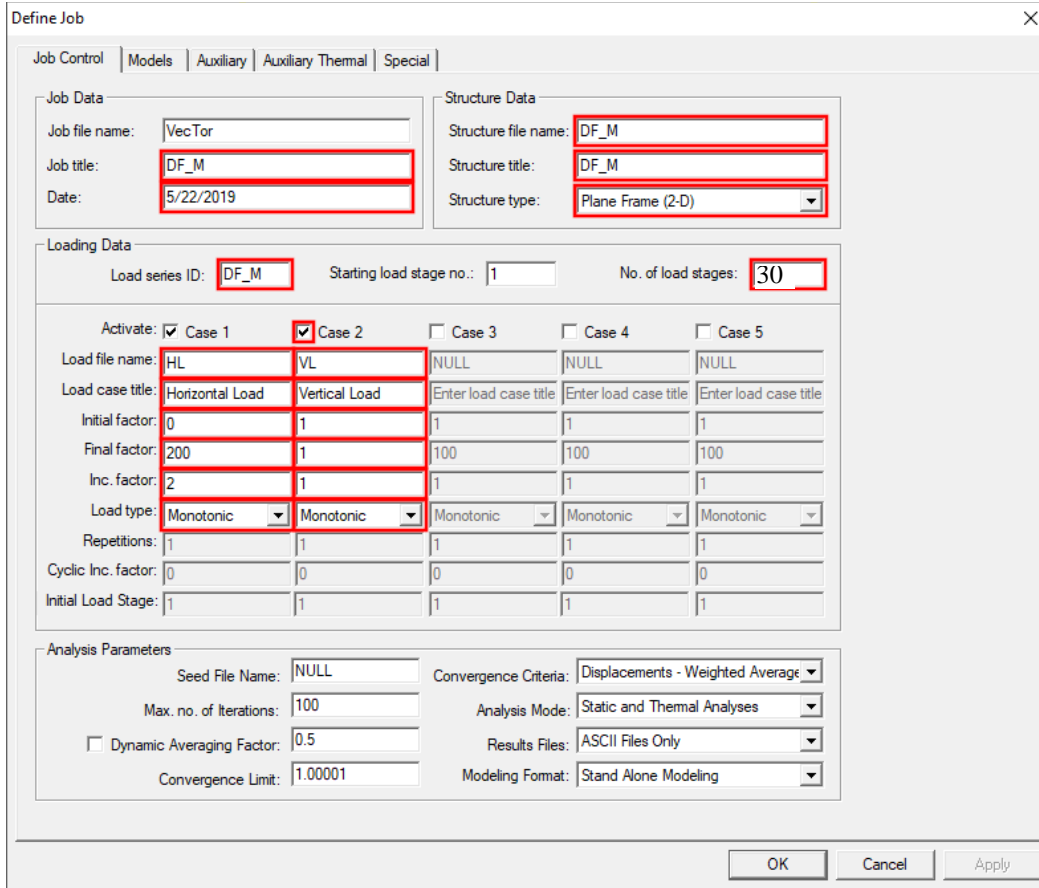


Figure 2 – “Define Job” window in FormWorks-Plus with all settings that are changed for the Duong Frame from the default to the settings shown with a red box around them.

After all of the information has been input, press the OK button to save the changes made. Save the Job file using the Red save icon displayed in **Figure 3**. If this button is grayed out, the project in FormWorks-Plus still needs to be saved.

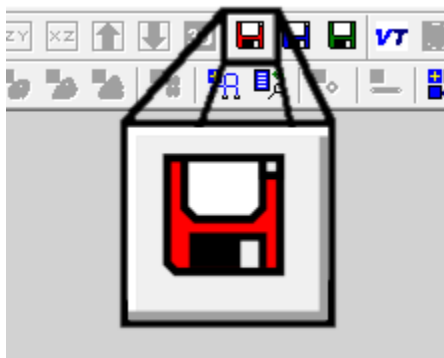


Figure 3 – “Save Job File” button.

3. DEFINING MATERIALS AND SECTIONS

The next step is to define the materials and cross-section properties. Open the “Define General Material Properties” window by pressing the button displayed in **Figure 4**.

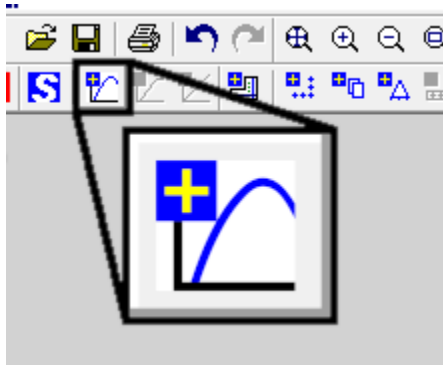


Figure 4 – “Define Materials” button.

In the first window, displayed in **Figure 5**, create all cross-sections used within the frame. Creating the cross-sections with doubled rebar at the intersections of columns and beams will be done later. In the case of the Duong Frame, six different cross-sections are defined, one of which was for doubled rebar (i.e., the frame joints). Each of these different cross-sections is given custom colors and names so that when the frame is defined, it will be easier to identify the type of cross-section for each member.

Figure 5 – First window from the “Define Materials” section where all of the concrete and stirrup properties are entered, where the color represents the material being defined.

After all of the different cross-section types are input, press the next button to go on to the “Define Concrete Layer Properties” window, seen in **Figure 6**. In this window, the option for “Auto Layer Generation” can be used for each cross-section but the results should be checked to ensure accuracy. Once this has been done for all cross-sections or manually entered for each cross-section, ensure that any cross-section that is being used to double rebar at joints has twice the transverse and out of plane reinforcement. Once everything is filled out, press the next button.



Figure 6 – Second window of the “Define Materials” section where all of the details of the concrete layers are input or generated for each section.

The third and last window is the “Define Longitudinal Reinforcing Bar Layer Properties” window where all of the rebar layer information will be entered in for each cross-section. In this example, all of the rebar information will be entered manually because the rebar is not evenly spaced in most sections. An image of what a section should look like in this window after having completed all of these steps can be seen in **Figure 7**. For any cross-sections where the rebar is to be doubled, enter in twice the number of bars. Once finished, press the done button to ensure that all changes were kept.

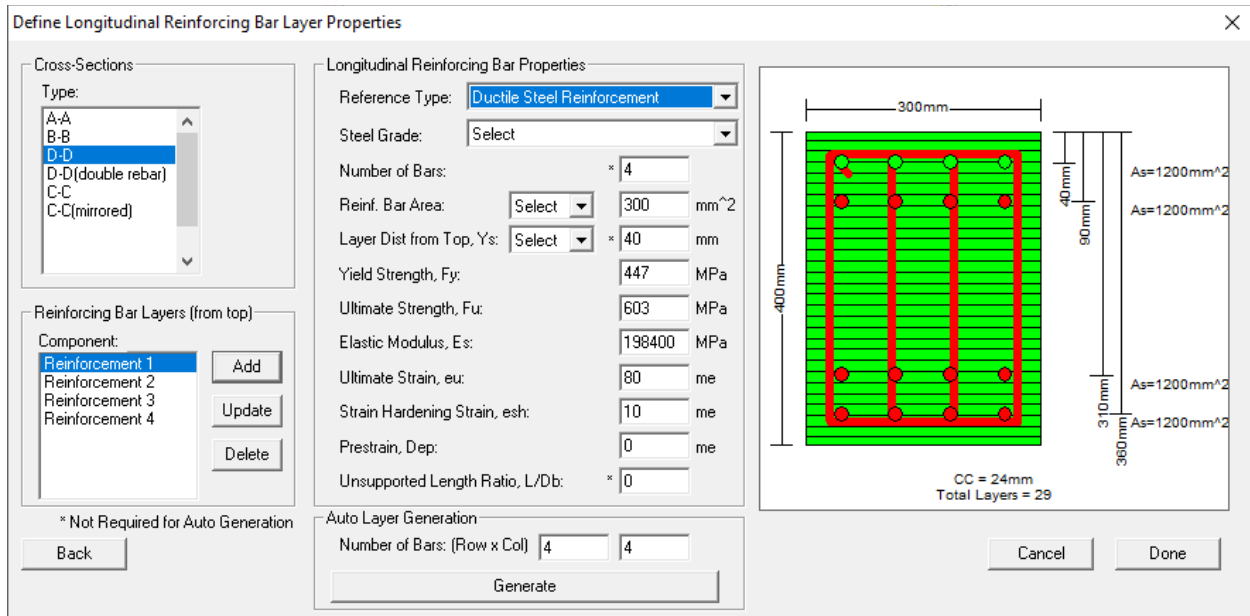


Figure 7 – Last window of the “Define Materials” section where all of the rebar layers are defined for each cross-section.

4. MESHING THE FRAME

Press “Define and Mesh Structure”, and open the window using the button shown in **Figure 8**.



Figure 8 – “Define and Mesh Structure” button.

Within the window that opens Layout the start and endpoints of the different material segments. Be sure to use the “Double Rebar at Joints” option at the bottom of the window. For the Dought Frame, the rebar needed to be doubled at the base of the frame as well as at the intersection points. FormWorks-Plus can only double rebar at joints using the “Double Rebar at Joints” option so the doubling of rebar at the bottom of the frame had to be done manually. Do this by creating regions at the bottom of the frame to double the rebar in and apply the cross-section with the doubled rebar

that was prepared earlier. The window with the inputs for the Doung Frame and the graphical representation of the information can be seen in **Figure 9**.

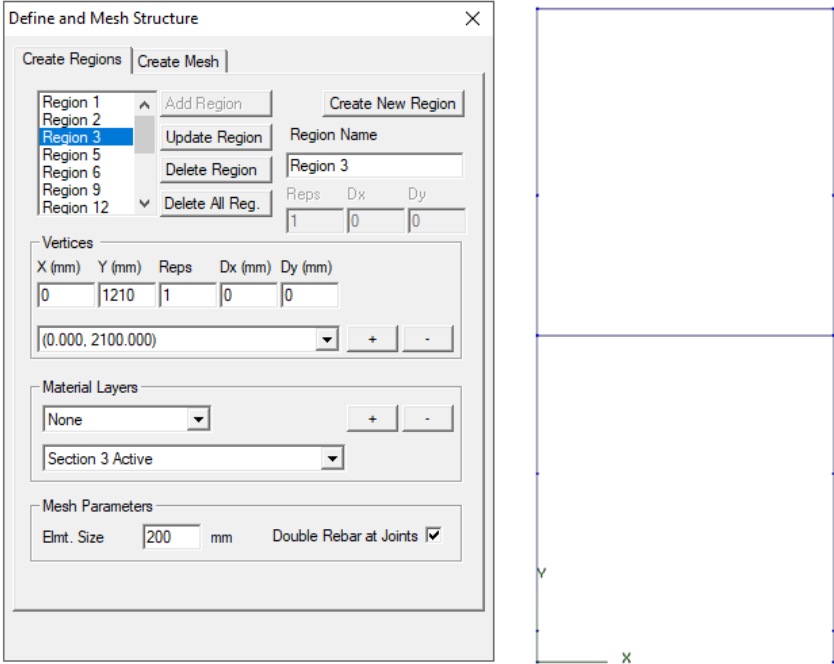


Figure 9 – “Define and Mesh Structure” window filled out with an overview of what is entered.

After defining all of the segments that make up the frame and giving them their material types, switch to the “Create Mesh” tab. This tab can be seen in **Figure 10**.

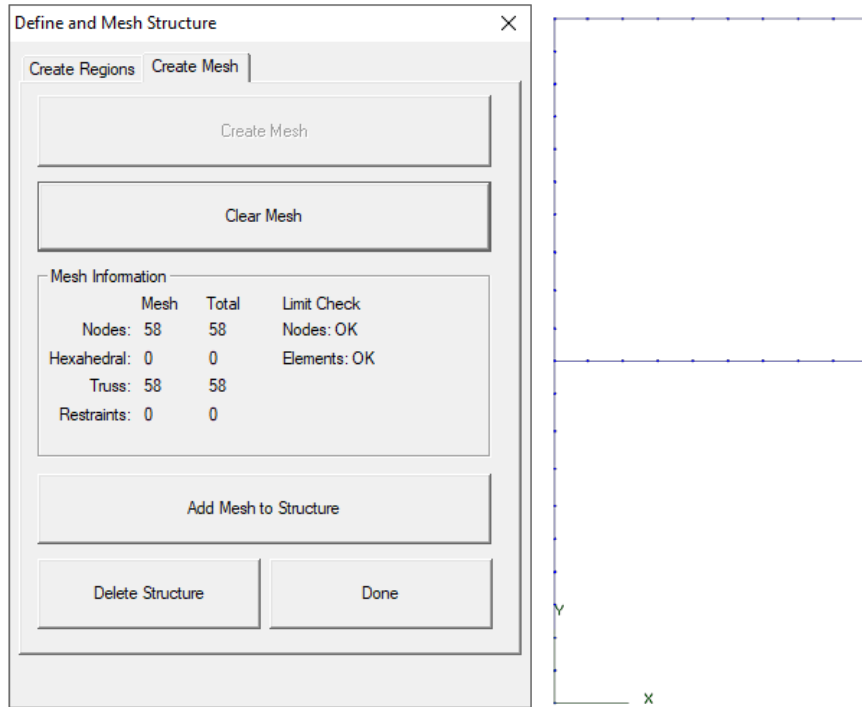


Figure 10 – “Define and Mesh Structure” window, with an overview of the mesh.

On this tab use the “Clear Mesh” button to ensure that there are no unwanted points. Use the “Create Mesh” button to create a mesh based on the inputs in the previous tab. Once this is done, a simple wireframe with all of the nodes will be displayed in the main window. If the frame looks right, press the “Add Mesh to Structure” button to confirm the auto-generated frame. Once the auto-generated frame is confirmed, it should take on the color of the material segments that define it. The meshed Doung Frame can be seen in **Figure 11**.

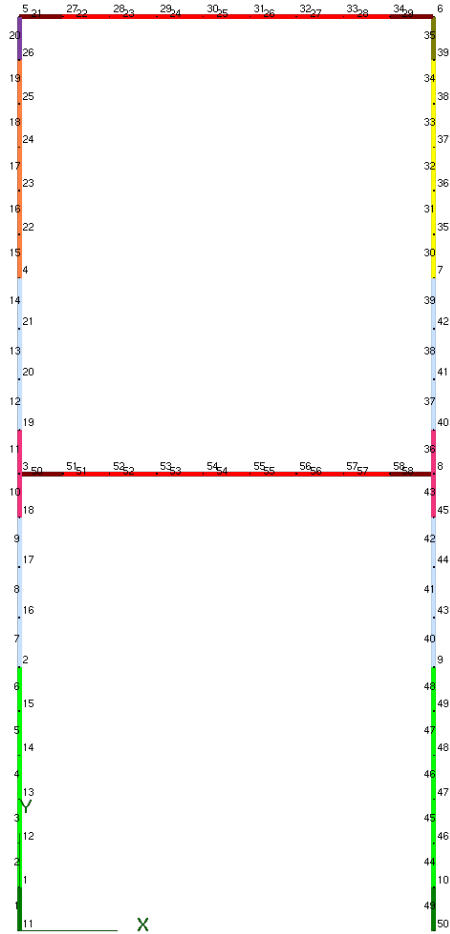


Figure 11 – Meshed frame from the “Define and Mesh Structure” window.

5. CONSTRAINING THE FRAME

Open the “Create Support Restraints” window by pressing the button shown in **Figure 12**.

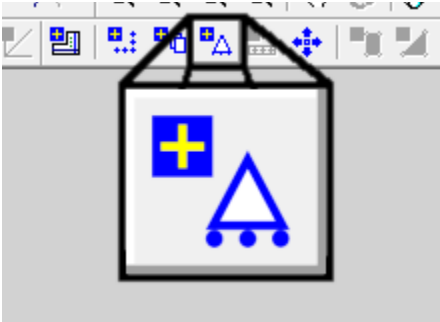


Figure 12 – “Create Support Restraints” button.

Enter in all of the constraints that are on the frame being modeled. After entering in each constraint, a graphical representation of that constraint should be shown in the FormWorks-Plus window, which can be seen in **Figure 13**.

Create Support Restraints

node	Restrain d.o.f.				# nodes	d node	# nodes	d node	# nodes	d node	Total								
1	<input checked="" type="checkbox"/> DX	<input checked="" type="checkbox"/> DY	<input checked="" type="checkbox"/> DZ	<input type="checkbox"/> R1	<input type="checkbox"/> R2	1	1	1	1	1	1	6							
<table border="1"> <tr> <td>1</td> <td>DX</td> <td>DY</td> <td>DZ</td> </tr> <tr> <td>55</td> <td>DX</td> <td>DY</td> <td>DZ</td> </tr> </table>												1	DX	DY	DZ	55	DX	DY	DZ
1	DX	DY	DZ																
55	DX	DY	DZ																

Selection Mode

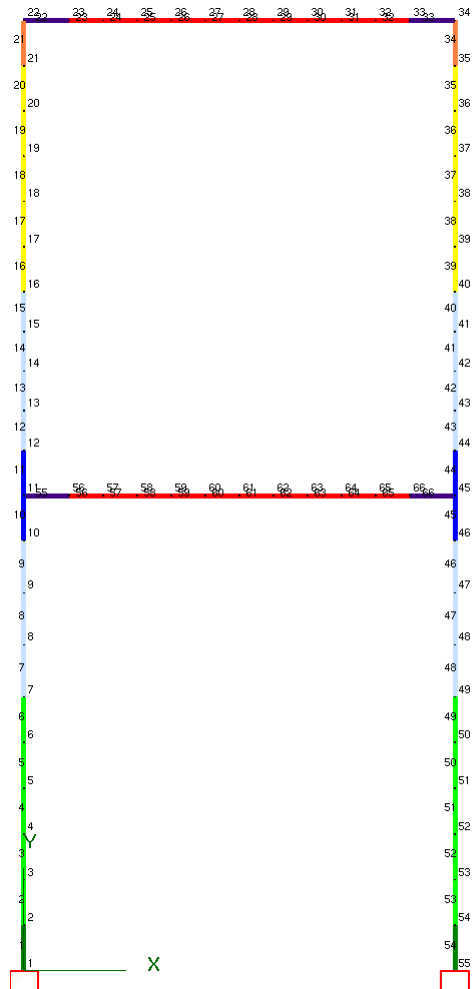


Figure 13 – “Create Support Restraints” window filled out and the meshed frame.

After all of the constraints are entered, press the done button on the window and save the structure using the button shown in **Figure 14**.

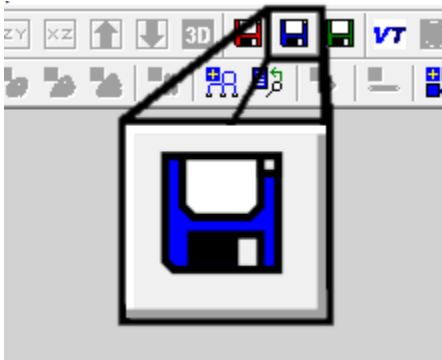


Figure 14 – “Save Structure File” button.

6. LOADING THE FRAME

The last step in modeling a frame is to define the load cases. First, ensure that the first load case is selected by using the button shown in **Figure 15**.



Figure 15 – “Load Case 1” button.

If the button is depressed, “Load Case 1” is selected. This load case corresponds with the earlier defined “Case 1” in the “Define Job” window. As decided earlier, this load case will be the controlled displacement load. Open the “Apply Support Displacements” window using the button in **Figure 16**.

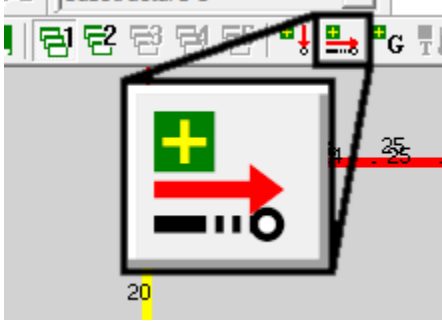


Figure 16 – “Support Displacements” button.

In this window, define the node/nodes that the controlled displacement is being applied to and fill the rest of the numbers. After adding the displacement, a red arrow will appear on the frame. Ensure that the arrow/arrows are facing the correct direction. Once the displacement force has been input correctly, press done and switch to “Load Case 2” using the button in **Figure 17**.

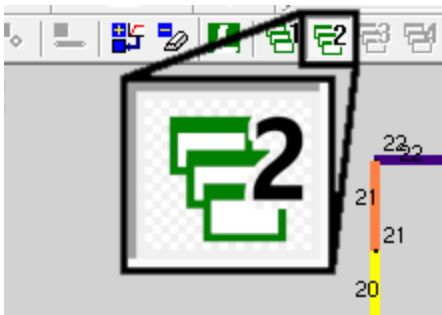


Figure 17 – “Load Case 2” button.

For the Dourg Frame, this load case will be a constant load case. To apply a load, open the “Apply Nodal Loads” window using the button in **Figure 18**.

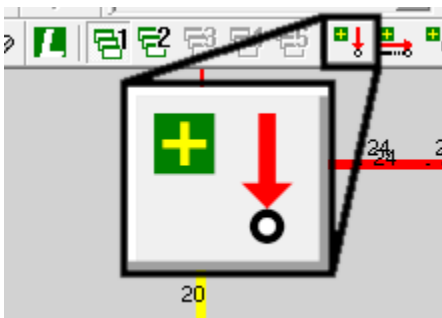


Figure 18 – “Nodal Loads” button.

Enter in the node of the point the constant load applied to the frame and its magnitude and press apply. Multiple loads can be applied in this way. For this Frame, two constant forces are added

under “Load Case 2”. There will be a red arrow for each load entered pointing the direction that it is applied. Once all of the constant loads are applied, press done. The last step is to save the load data file using the save button shown in **Figure 19**.

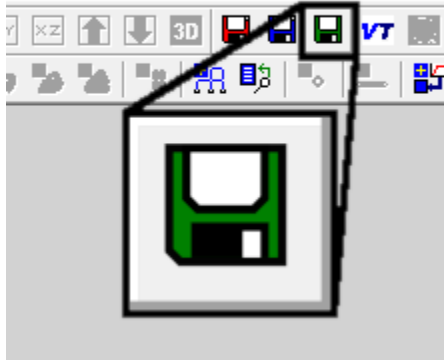


Figure 19 – “Save Load File” button.

7. RUNNING THE ANALYSIS AND OBTAINING THE RESULTS

At this point, we are ready to run VecTor5 which can be done using the button shown in **Figure 20**.

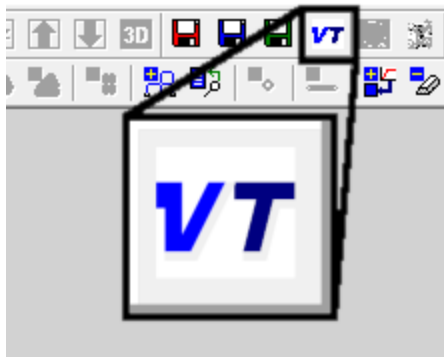


Figure 20 – “Run VecTor Processor” button.

Ensure that the project is saved often to avoid losing your work. After running VecTor5, following the same steps in [User Bulletin 4](#), use Janus to visualize the results from VecTor5. **Figure 21** shows the Doung Frame analysis results in Janus at load stage 14. For more information on how to use Janus, please refer to User [Manual of Janus](#).

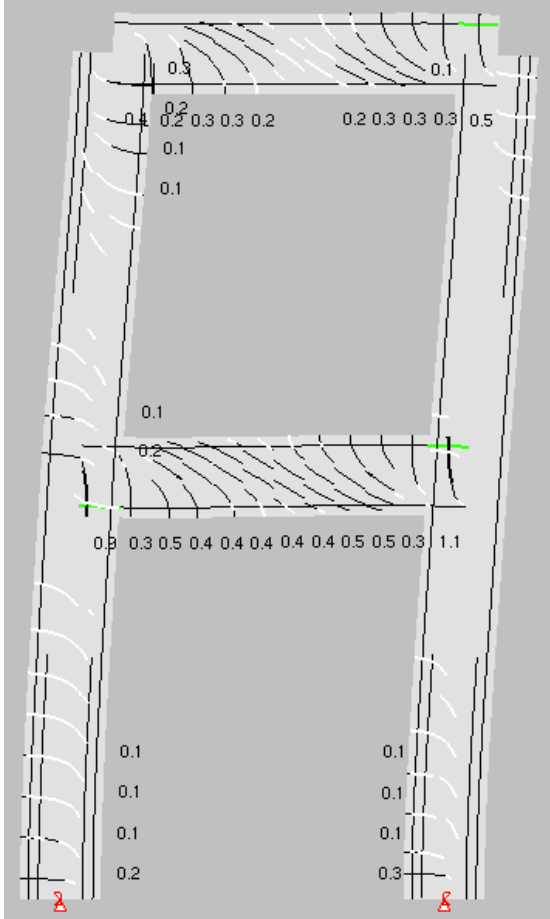


Figure 21 – Load stage 14 of the Doung Frame output in Janus.

8. REFERENCES

Guner, S. and Vecchio, F. J. (2008), “User’s Manual of VecTor5,” Online Publication, 88 pp.

[Download](#) (see User’s Manuals tab).

Duong, K. V., Sheikh, S. A., and Vecchio, F. J. (2007), “Seismic Behavior of Shear-Critical Reinforced Concrete Frame: Experimental Investigation,” ACI Structural Journal, pp. 304-313.

[Download](#)