USER'S MANUAL OF JANUS FOR VECTOR5

by

Agneya Sunil Loya

Daniela D. S. Lourenço

Serhan Guner

Frank J. Vecchio

March 2017 - V2.0

TABLE OF CONTENTS

1. INTRODUCTION

Janus is a post-processor computer program for visualizing the analysis results of VecTor suite of programs. This manual is specific to its use for VecTor5 models. Janus presents information regarding the deformed shape of structure, crack propagation along with their widths and locations, rebar positions with stresses and sectional views of members.

The main purpose of this manual is to make user familiar about the program and how to use it. This draft is intended to be a comprehensive and practical manual, elaborating the input files and working on various models.

Janus is gradually developed and new features are added. The following guide briefly summarizes the currently available features for VecTor5 models. This manual, as well as other related publications, can be downloaded here.

2. LOADING A MODEL

Loading a model is very simple and straight forward. Open 'Janus.exe' and browse for your VecTor.JOB file using the File, Open button or the Open icon on the taskbar shown as 📂.

Janus displays loading of different model files with a pop-up window as shown in Figure 1;
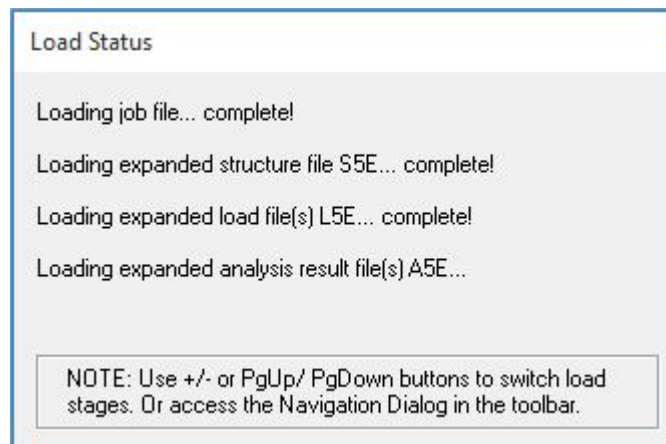

Figure 1: Pop-up window while loading Vector.Job file

New users should pay attention to the note shown in the above window. Load stages could be changed by:

1. +/- buttons

2. PgUp/ PgDown buttons

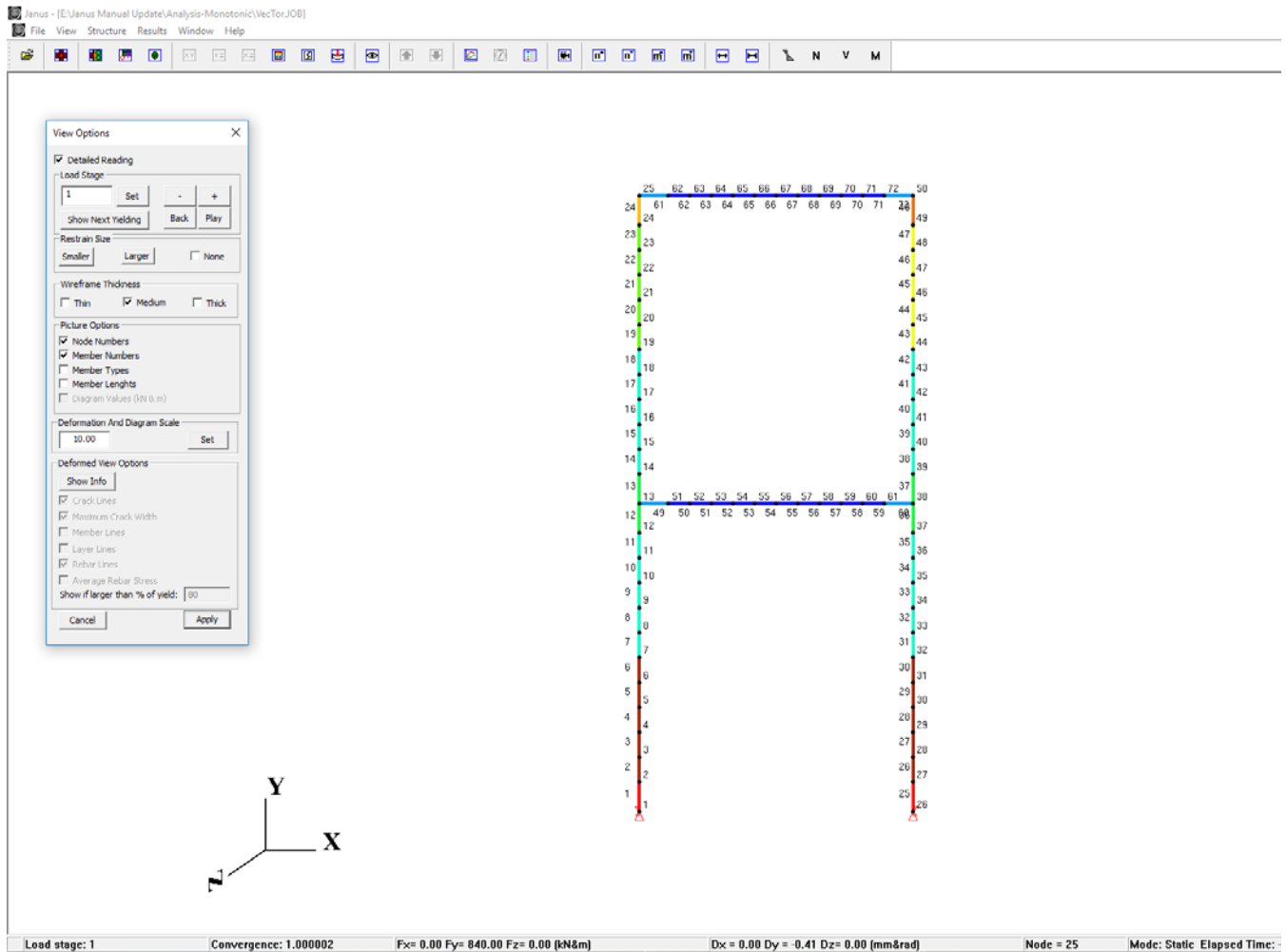The Figure 2 below shows the interface of the program once model is opened;

Figure 2: Interface of the Program

3. LAYOUT

All controls can be accessed through the taskbar on top. The status bar at the bottom contains indicators for (left to right):

1. Load stage number
2. Convergence factor value
3. Cumulative applied force (force controlled) OR cumulative reaction developed (displacement controlled)
4. Displacement of node that undergoes max displacement
5. Number of node that undergoes max displacement
6. Mode of the Analysis of structure
7. Time elapsed in Analyzing the structure.

The VT5 View Options dialog opens by default on startup and contains the most important view controls for visualizing the model. It can be accessed using the 'eye button' in the taskbar.

4. THE TASKBAR

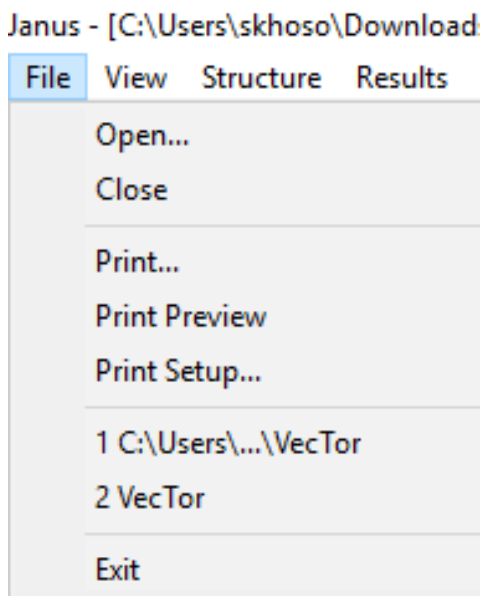## 4.1 MENUS

4.1.1 File menu

Figure 3: File menu with different options

The file menu shown in Figure 3, contains standard options to **Open** and **Close** files, **Print** the screen, **Open Recent** files and **Exit** the application.
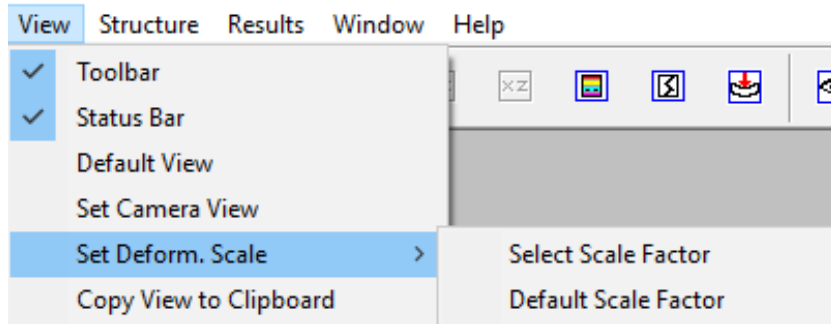
4.1.2   **View menu**

Figure 4: View menu with different options

The **Toolbar** and **Status Bar** are active by default and could be hidden by unchecking them using view menu shown in Figure 4.

The **Default View Option** sets the model back to the view it initialized/displaced in.

Set Camera View provides controls to render the view from different angles and zoom levels. The only relevant control that need to be known is the **Set Zoom Scale factor** which allows finer or coarser control over the zooming in and zooming out of the model.

Set Deformation Scale has a submenu with options to enter a user defined value for the factor by which deformations are scaled and a button for the **Default Scale Factor** which is set to 10.0.

Copy View to Clipboard is for copying a snapshot of the screen to the clipboard. However, this button has bugs and the it is recommended to use the default 'Snipping tool' provided by Windows.
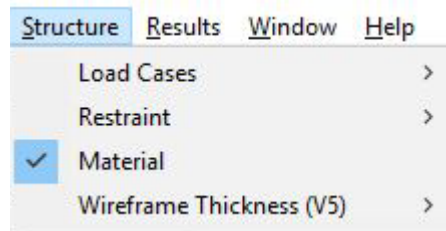
4.1.3 Structure menu

Figure 5: Structure menu with different options

Structure menu contains controls either to show or not the load cases, restraints, material colors and set the wireframe model thickness. However, the View Options dialog described later covers all these controls in a much more condensed and neat manner.
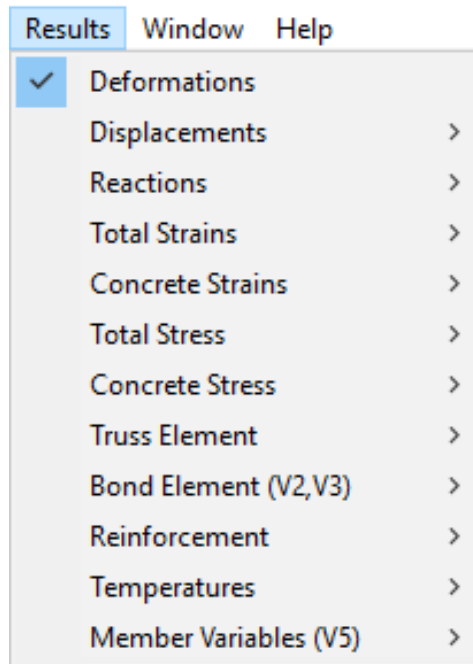
4.1.4 Results menu

Figure 6: Results menu with different options

Table 1: Function of each option shown in Results menu

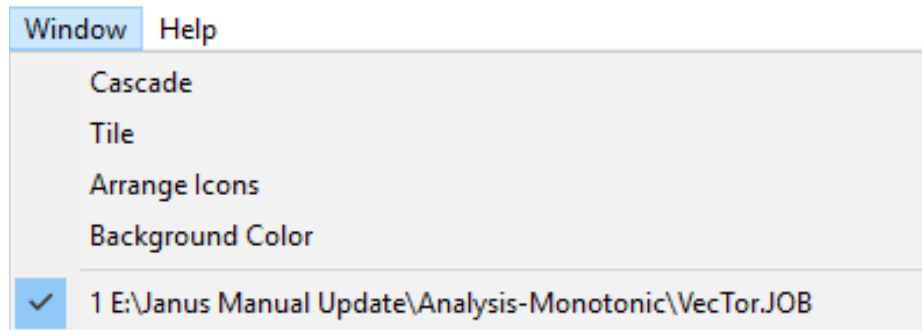| Button	Function
Deformations	toggles the deformation of the model on and off
Displacements	x, y and z displacement gradients throughout the model
Reactions	x, y and z displacement gradients throughout the model
Total Strains	principal major and minor strain gradients
Concrete Strains	concrete strain gradients
Total Stress	**Not applicable for VecTor5**
Concrete Stress	Concrete stress gradients
Truss Element	**Not applicable for VecTor5**
Bond Element	**Not applicable for VecTor5**
Reinforcement	Longitudinal and transverse reinforcement strain and stress gradients
Temperatures	**Not applicable for VecTor5**
Member Variables	Member forces and member deformations

4.1.5 Window menu

Figure 7: Window menu showing different options

Multiple windows could be open on same screen for different projects using **Cascade** and **Tile** options to position the windows.

Arrange Icons is a redundant button:

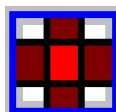**Background Color** option allows the to change the background color of the model view screen.

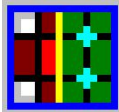### 4.2 TOOLBAR BUTTONS

4.2.1 Open model file button

This icon allows the to open an existing model by putting Vector.JOB file.

4.2.2 Hotspot view button

The hotspot view gives controls to view various structural parameters such as deformations, stresses, strains and cracks as gradients on the model within a user defined range of values.

4.2.3 Material view button

This button shows or hides colors to indicate different material types.
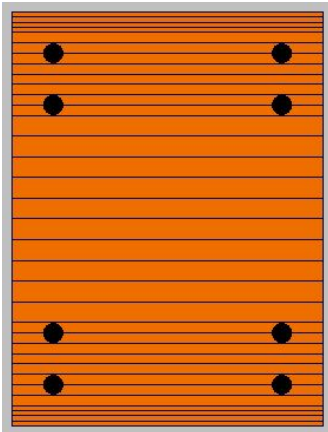
4.2.4 Two-Dimensional view button

This option helps to view different cross-sections of the frame. Once the cross-sectional view displayed, the same option could be used to switch back to the original frame view as initialized.

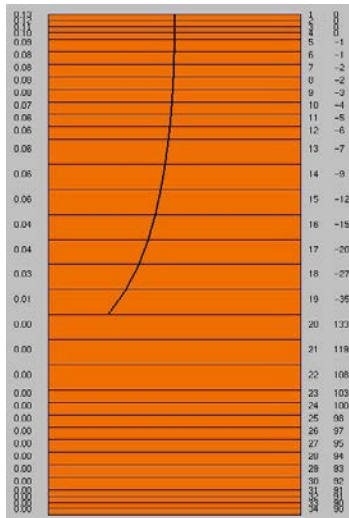### 4.2.5 Sectional layer view button

The sectional view button allows the to select any member type of the frame to see its cross-sectional layer profile along with the reinforcement bar layout. An example of the sectional layer view is shown below:

4.2.6 Sectional crack view button

The sectional crack view button allows to select a member to see the cross-sectional layer profile along with the location of cracking in the member. A sample of this view is shown below. The list of numbers to the left of the layers is the value of crack width in each layer in millimeters, whereas the total number of concrete layers is shown on right side of the cross-section. The crack width and location changes in response to changing load stages.

4.2.7 Global frame deformation view button

The frame deformation view shows the 2-dimensional profile of the frame considering deformations, cracking and reinforcement bar stresses. Figure 8 shows the screen capture of a sample view of a model in this mode;

The numbers on various locations show crack width in millimeters whereas different cracks and various locations are shown in red, black and white colored lines.
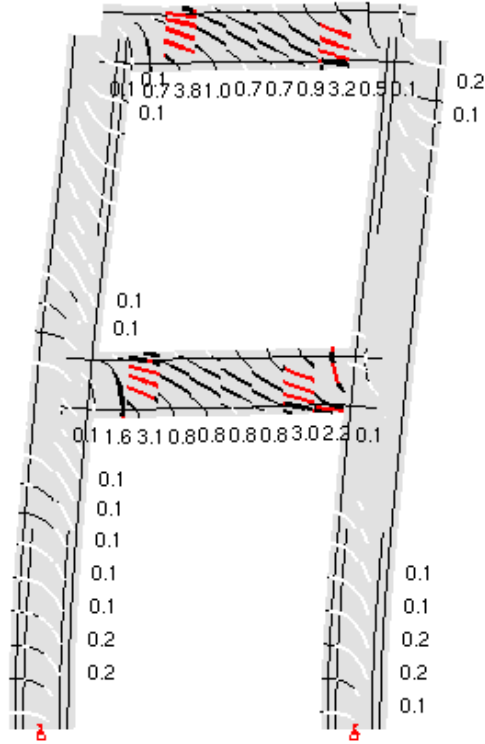
Figure 8: Deformed shape of frame with cracks and reinforcement bar stresses

a) Deformations

The frame undergoes larger deformation once load stages are increased. The frame could bring back to the un-deformed shape by clicking the Deformations option under the Results tab shown in Figure 9.
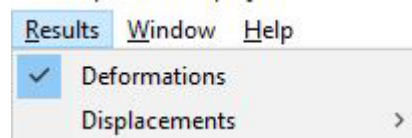

Figure 9: Results menu showing deformations and Displacement options

The calculated deformations are scaled to aid visualization. The default scaling factor for deformations is set to 10.0 and can be changed in the View Options dialog box.
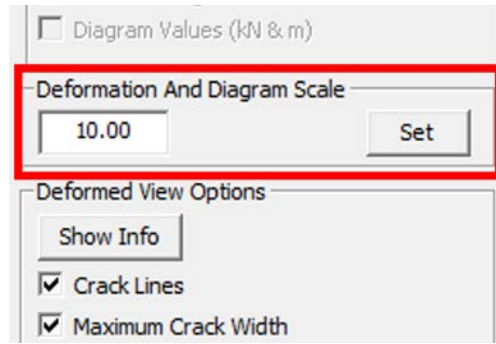
Figure 10: Deformations and Diagram Scale

b) Cracks

Cracks along with their width are displaced in various colors depending on their computed value. The detail of cracks developed during different load stages is given in Table 2.

Table 2: Detail of various color of cracks along with their widths

Crack color and thickness	True physical crack width range
White + 2 pixels thick	< 0.10
Black + 1 pixel thick	0.10 – 0.50
Black + 2 pixels thick	0.50 – 1.00
Black + 3 pixels thick	1.00 – 1.50
Red + 2 pixels thick	1.50 – 3.00
Red + 3 pixels thick	> 3.00

The legend shown in Figure 11 could be accessed by clicking 2D View option present in the task bar, where crack width is indicated with different colors and thickness.
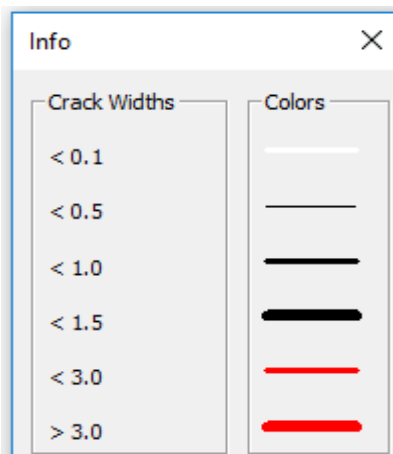

Figure 11: Legend showing crack widths along with their color

The maximum crack widths per member are also shown in Figure 8 above on the deformed frame. The cracks on various locations and the crack width could be displayed and hidden using the View Options box.

Maximum crack width and the crack lines could be accessed from the options shown in Figure 12. Deformed shape and increase in crack widths could also be accessed as indicated. **Show Next Yielding** option below displaces the successive bar yielding one after another.
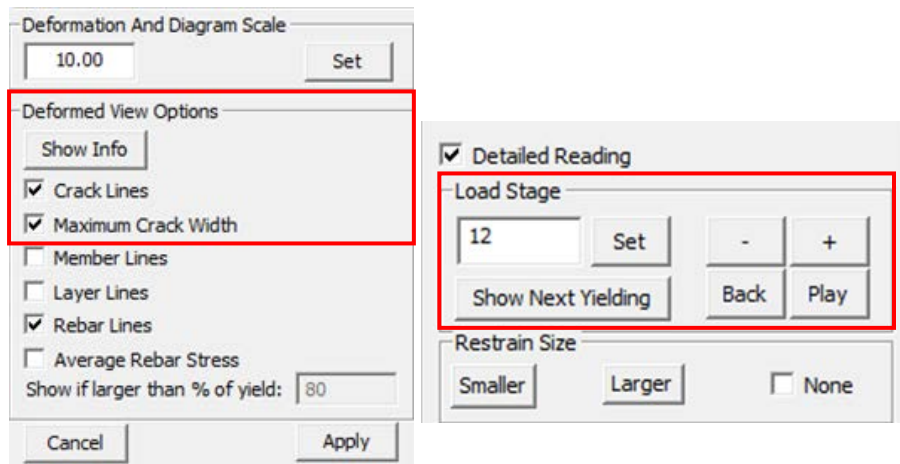

Figure 12: Windows showing deformed view options and different load stages

c) Reinforcement bars and stresses

Reinforcement bars are displaced once the frame is opened in the program. The colour of reinforcement bars varies with their state. Table 3 summarises this feature:

Table 3: Detail of rebars going through different states during analysis

Rebar color	Rebar state
Black	Normal
Green	Yielding
Yellow	Strain hardening
Continuous Red	20% before rupture
Dashed Red	Post rupture

The legend in Figure 13 could be accessed by clicking 2D View option present in the task bar, where Yielding, Strain Hardening, Before Rupture and Ruptured bars are shown with different colors and lines.
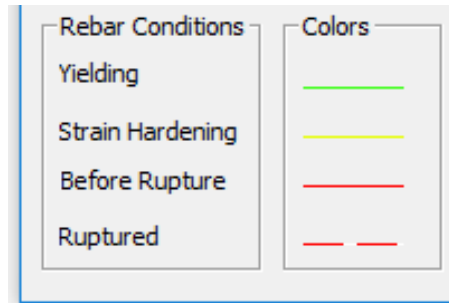
Figure 13: Rebar conditions indicated with different colors

Reinforcing barsand their stress values can be shown or hidden using the View Options box. Rebar stress values are shown only when they are above a certain percentage of their yield stress values. This percentage is set as 80% by default and can be altered in the View Options box shown in Figure 14.
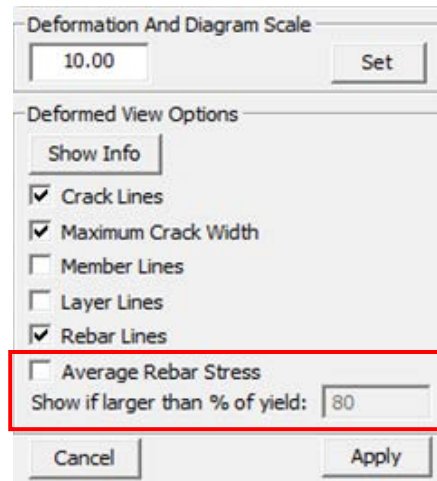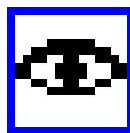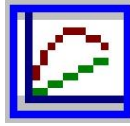

Figure 14: Detail of average Rebar stresses

4.2.8 View Options button

This button opens the View Options dialog box. The dialog box is open by default. Details on the options contained in the dialog box are in section.

4.2.9 Plotting Controls Button

The plotting controls button opens dialog boxes that allows to choose variables for plotting. The first dialog box opened is shown in Figure 15.
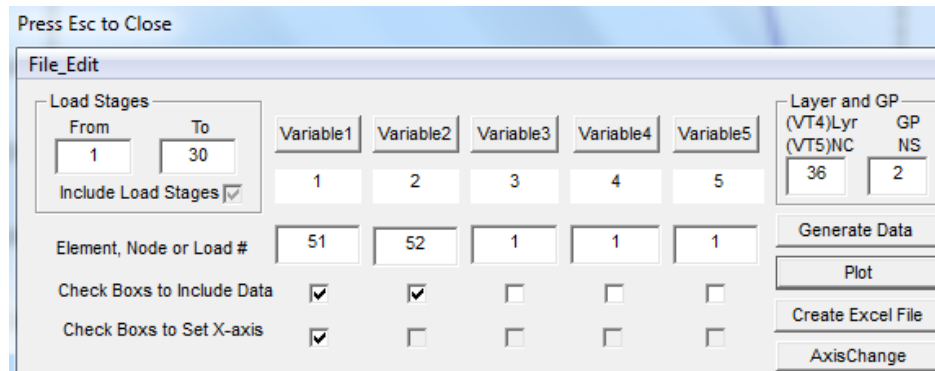

Figure 15: Dialog box with details to be entered for plot

To plot a graph:

1. Enter range of load stages
2. Select variables for plotting (usually 2 but could use up to 5)
3. Select the element (member) or node number for which the variables are required
4. Always check '*Check Boxes to Include Data*' corresponding to selected variable
5. Choose the dependant variable to be on the axis by checking the box '*Check boxes to Set X-axis*'
6. Select NC (concrete layer number) at the top right and/or NS (steel layer number) for layer specific variables
7. Click plot
8. The plots created are not always very clear as this feature is not completely ready. Clicking '*Generate Data*' generates the variable data for the load stage range in the white space below. This can be used to create your own graph in excel or a similar software.

4.2.10 Legend Dialog button

Legend dialog button shows material used along the frame with different colours The detailed information is shown in Figure 16 having concrete material. This button could be hide and shown by clicking on it.
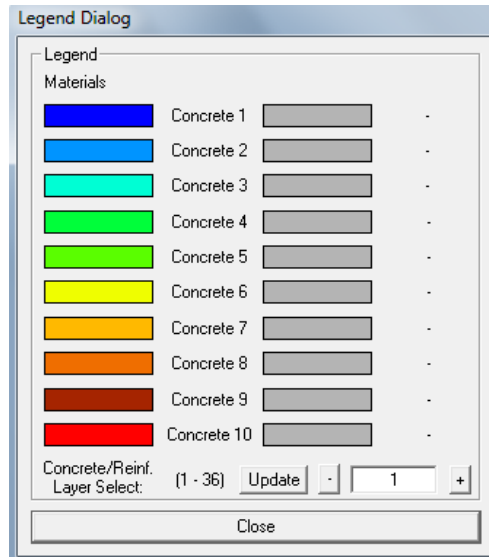
Figure 16: Legend Dialog box showing material with different colors

4.2.11 Video Dialog button

This option is currently unavailable for Vector 5 files in current version of Janus.
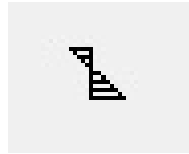
4.2.12 Text position controls

The **'n up'** and **'n down'** options shown, move the node numbers away or close from their respective node positions. The **'m up'** and **'m down'** options move the member numbers, member lengths, member types, crack widths and rebar stresses away or close to the model.

4.2.13 Footer width controls

These buttons control the width of various status indicators in the footer. The footer is sized differently on screens with different resolutions and in some cases the data overlaps or spreads outside the frame. Footer can be resized as per convenience ensuring that the same view shown below is displayed;

| Load stage: 17 | Convergence: 1.000314 | Fx= 344.510 kN Fy= 0.000 kN | Dx = 32.000 mm Dy = 0.000 mm | Node = 23 |

4.2.14 Plots option

The plot option displays another screen where different graphs related to the model could be selected. Since Janus consumes a large amount of memory for loading the information, the plot function may take some time, or crash if the memory limits are exceeded
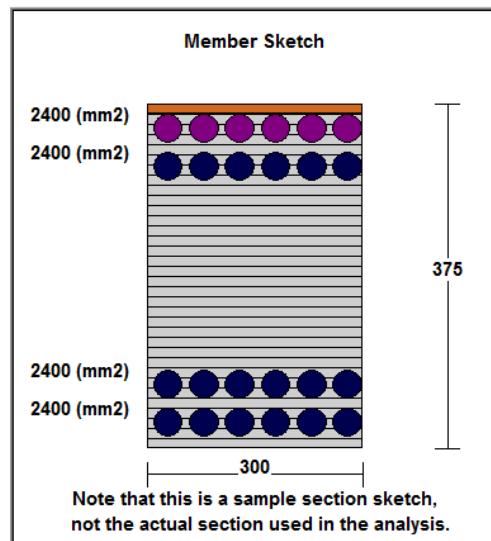

Figure 17: Rectangular Cross-Section of Beam

Janus always displays a rectangular cross-section shown in Figure 17, since it was developed to display the above-mentioned cross-section. The program provides an equivalent cross-sectional area as rectangular, even if the real section input is not rectangular.
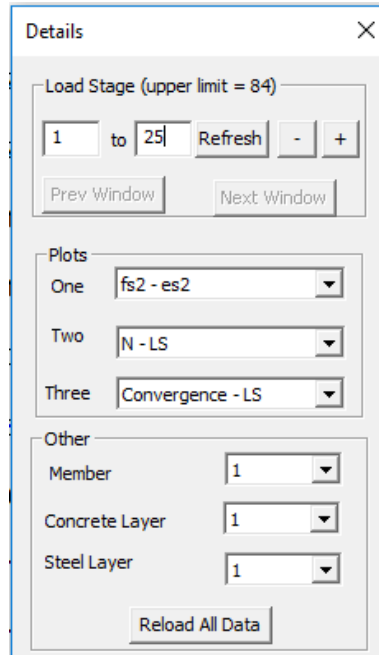
Figure 18: Dialog Box to provide details for plots

Each Plot could be accessed by providing details in the dialog box shown in Figure 8. Plot types, member details, concrete and steel layers could easily be selected from drop down arrows shown.
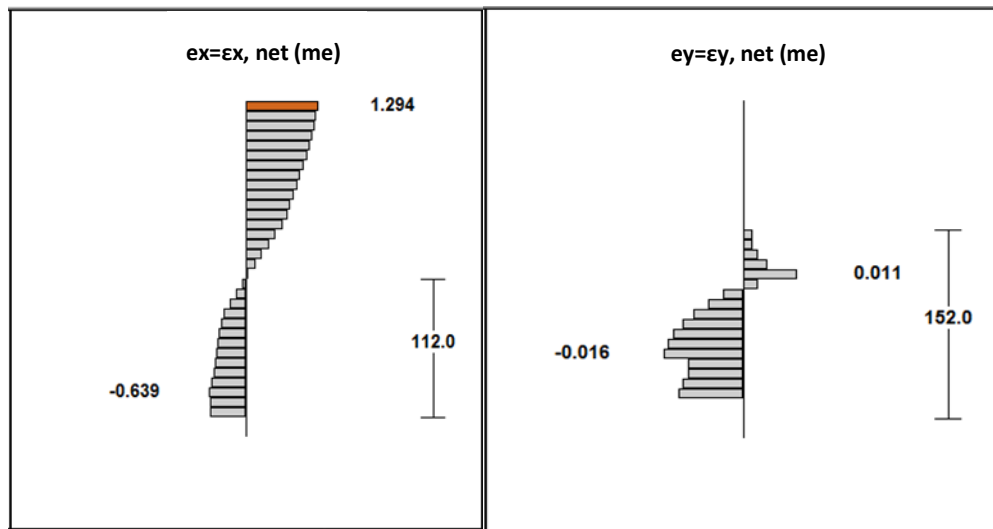

Figure 19: Strain diagrams (a) Longitudinal strains (b) Transvers strains

Figure 19 and 20 show details diagrams with different concrete layers. Peak positive and negative values in each layer are indicated. The total height of concrete layers is displayed on right corner of each diagram. Orange colour in diagram shows the values at selected concrete layer.
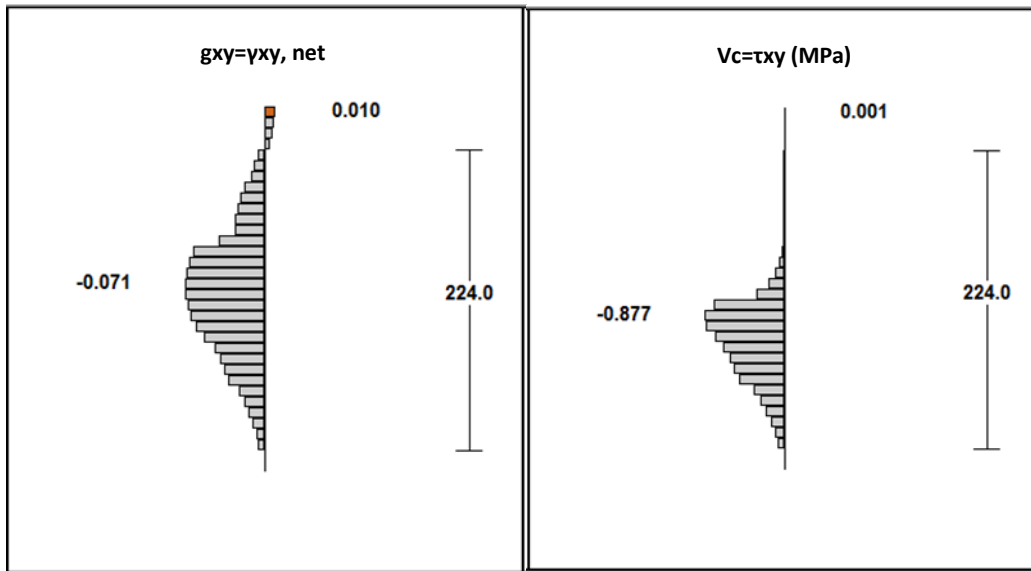
Figure 20: diagrams (a) Shear Strain (b) Shear Stress

Figures 21, 22 and 23 shows different graphs of Axial load vs load stages, stress vs strain and convergence vs load stages respectively. The shown plots could be accessed by providing details in the dialog box shown in Figure 18.
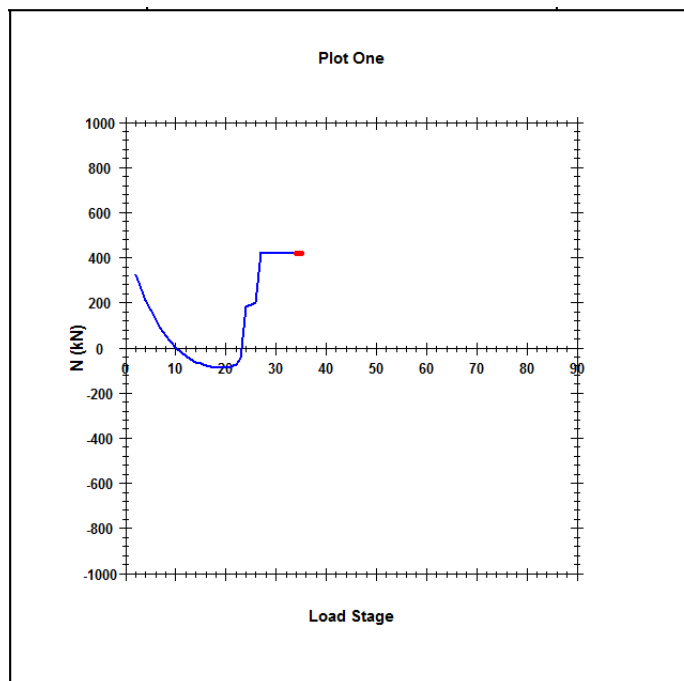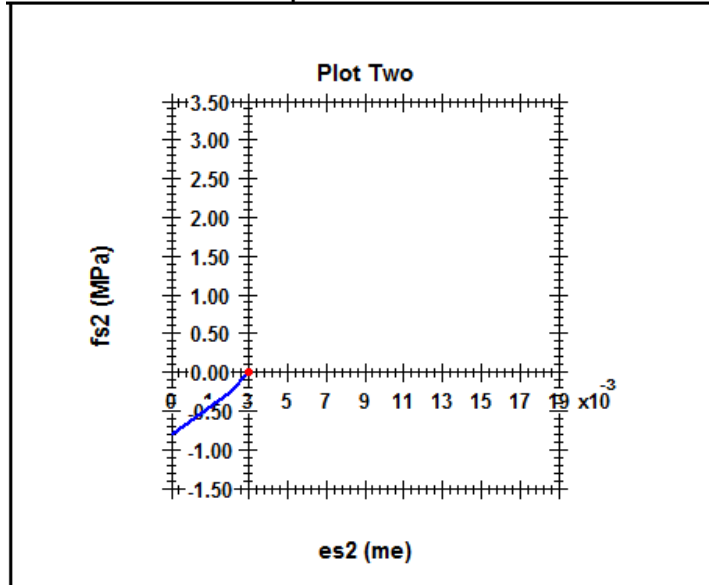

Figure 21: Axial Load vs Load Stages Graph
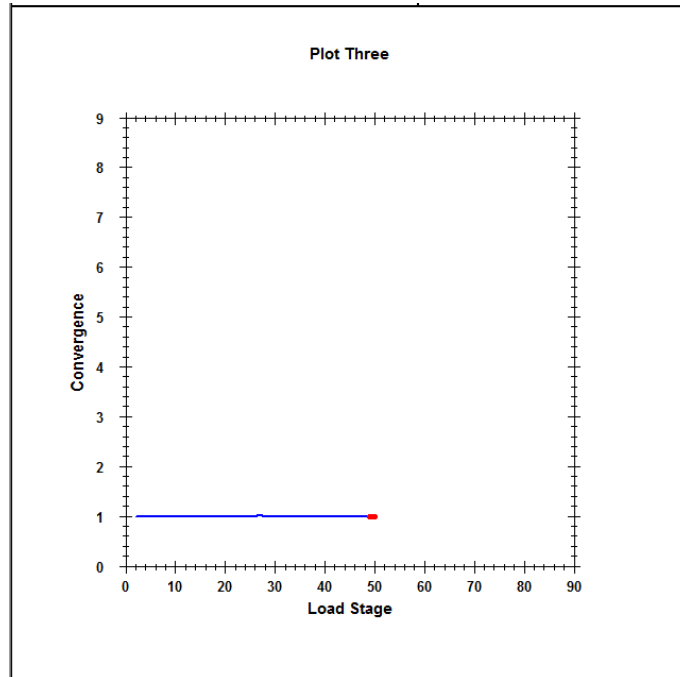
19

Figure 22: Stress vs Strain Graph

Figure 23: Convergence vs Load Stage Graph

4.2.15 Axial Diagram button

N

This option displays the axial diagram and reactions both in X and Y direction as well as moment in Z direction for each load stage. The positive values are shown with pink colour whereas blue colour shows negative values.The valuescan be shown and by checking and unchecking the Diagram Values box in the View Option menu shown in Figure 24.

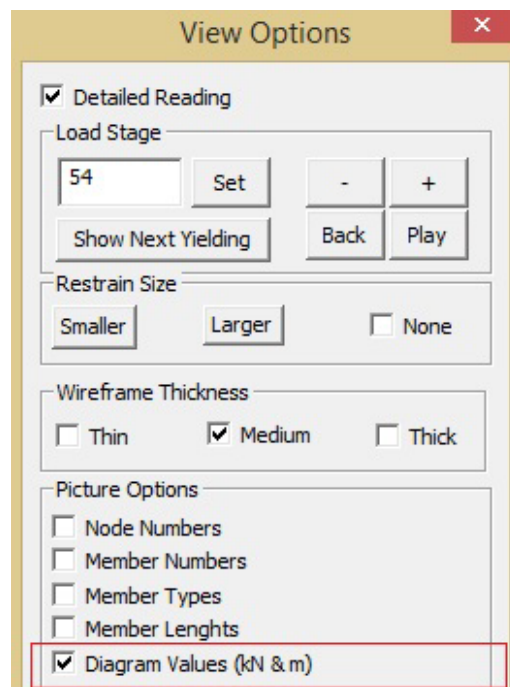**Figure 24:** Dialog box showing highlighted option for values on Diagram in (kN and m)

An example of Axial Force Diagram is shown in Figure 25. This diagram can be accessed by checking the Diagram Values option shown in View options menu. The forces and reactions are shown with single head arrow, whereas moment is shown with two headed arrows.
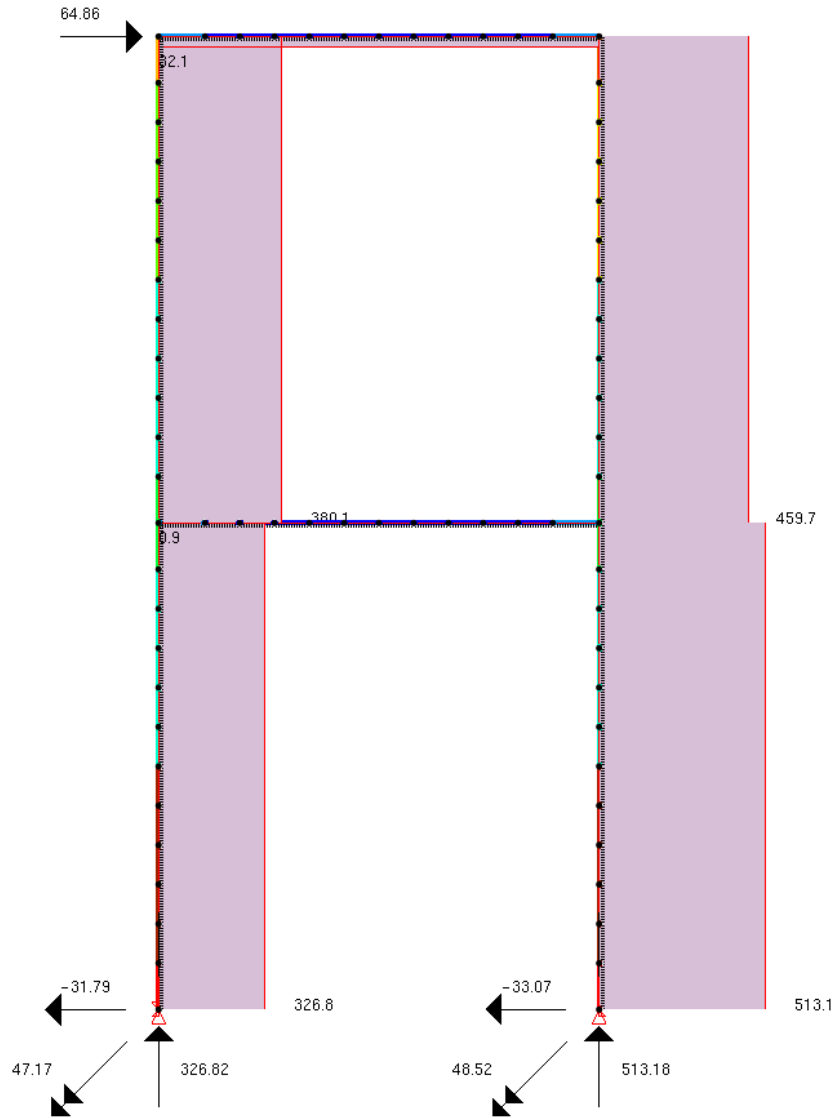
Figure 25: Axial Force Diagram (kN and m)

4.2.16 Shear Diagram control

For the shear diagram, a recommendation would be to increase the Deformation and Scale in the View Option menu shown in Figure 26. This way, better understanding of the behaviour of structure on each load stage could be observed.
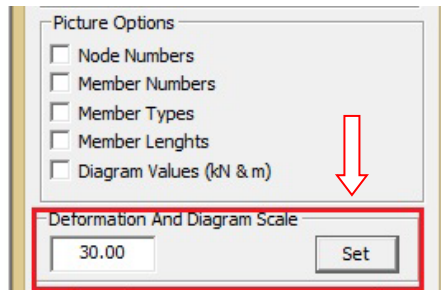
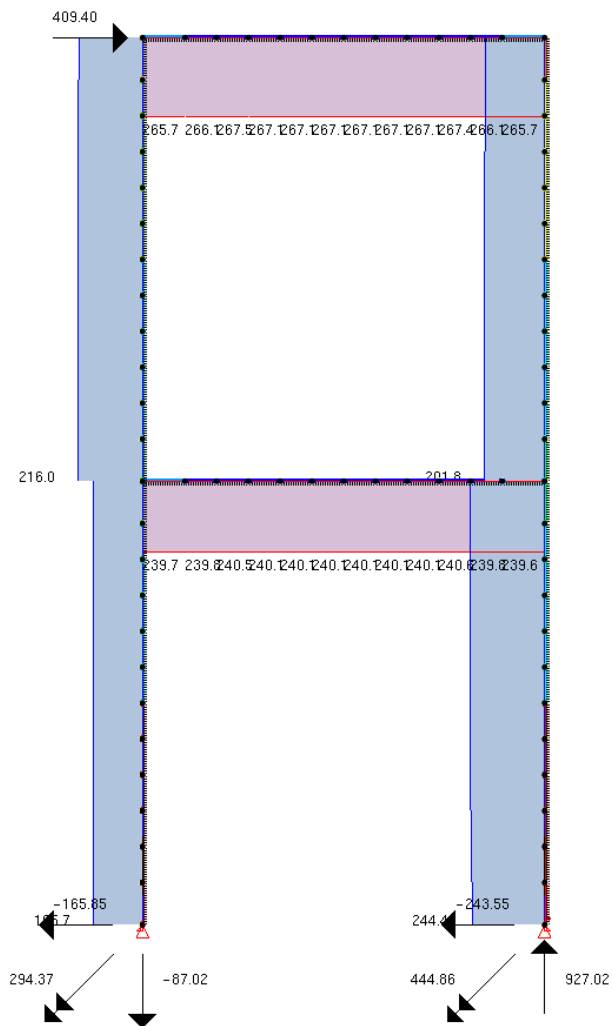**Figure 26:** Deformation and Diagram Scale

Figure 27: Shear Force Diagram (kN and m)

The forces and reactions are shown with single head arrow, whereas two headed arrows show moment.

4.2.17 Moment Diagram control

M

This option helps to control the scale and can also show and hide the values for the Moment Diagram in the similar way as it was done for Axial and Shear diagrams. The moment diagram could be both linear and parabolic depending upon the loading conditions.
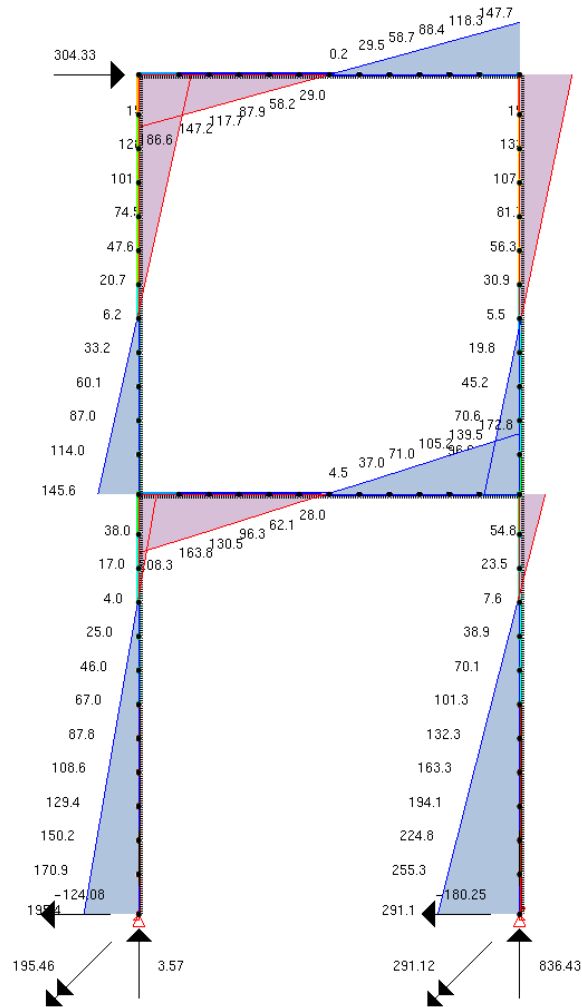

Figure 27: Bending Moment Diagram (kN-m)

The forces and reactions are shown with single head arrow, whereas two headed arrows show moment.

5. VIEW OPTIONS DIALOG BOX

Behaviour of frame while undergoing different load stages could be observed by clicking the Show Next Yielding or Play button shown in Figure 29, which will display structure until its complete failure. The first yielding in rebar could be observed in Figure 28 (a) whereas next yielding is shown in Figure 28 (b). When undergoing any changes a user must click on the property "Set" button or hit the Apply button at the very bottom of the menu. Detailed explanation of each button in the View option dialog box is given in Table 4.
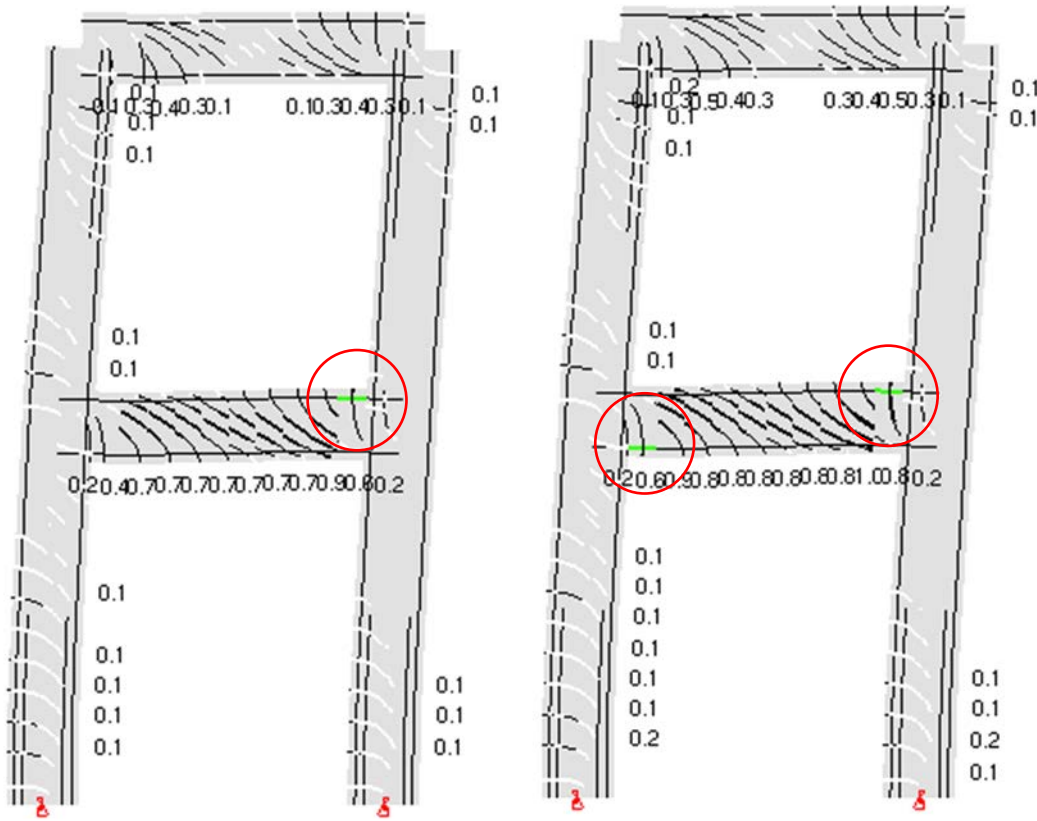

Figure 28: Deformed shape of frame (a) yielding of one rebar (b) yielding of two rebars
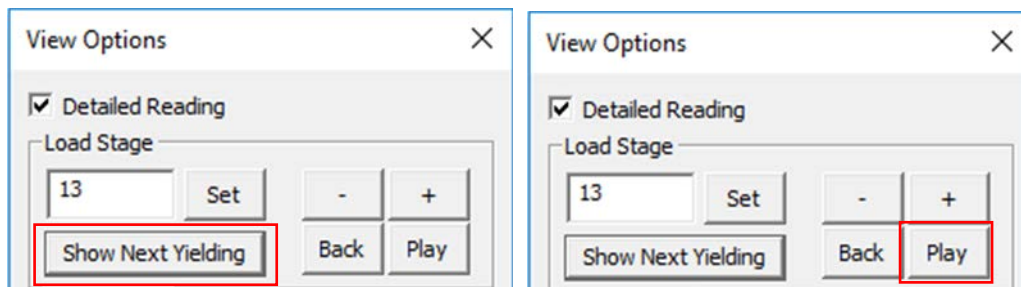

Figure 29: View option dialog box highlighted with (a) Show Next Yielding (b) Play

Table 4: Detailed explanation of each button in View option dialog box

Button/ feature	Function
Detailed reading	Enables reading of detailed member outputs. Janus is fast enough to handle large models and this option can be left checked.
Load Stage	Shows current load stage.
Set Load Stage	Helps to jump back or forward to a specific load stage.
Plus/Minus	Go one load stage up/down.
Play/Back	Automatically plays the structure increasing/decreasing load stages
Show Next Yielding	Turns the structure into a 2D model and goes until the next load stage of yielding
Restraint size	Restraints can be made smaller, larger or hidden
Wireframe thickness	Changes the thickness of the model in the line diagram view
Node numbers	Shows or hides node numbers
Member numbers	Shows or hides member numbers
Member types	Shows or hides member types abbreviated as MT
Member lengths	Shows or hides member lengths in mm
Diagram Values	Used to show or hide the diagram values for the members
Deformation and Diagram Scale	Displays current scale value and allows to set a new value. Enter a scale value between 0.1 and 100 and click 'Set'
Show Info	Opens a box with additional information about the deformed structure
Crack Lines	Shows or hides cracks
Rebar Lines	Shows or hides reinforcement bars
Max. Crack Width	Shows the maximum crack width per member in mm
Average Rebar Stresses	Shows the average rebar stress per member
Show if larger than _% of yield	Allows to specify cutoff value above which rebar stresses are shown
Member lines	Shows or hides member lines in frame view
Layer lines	Shows or hides layer lines in frame view

6. RESULTS MENU

This menu helps to see the results of the analyzed structure. Different parameters are shown in Figure 30 where concrete strains and stresses are highlighted. The legend shown in Figure 31 gives detail of the points of high and low concentration for both concrete strains and stresses. Example of deformed frame with these parameters are shown in Figure 32.
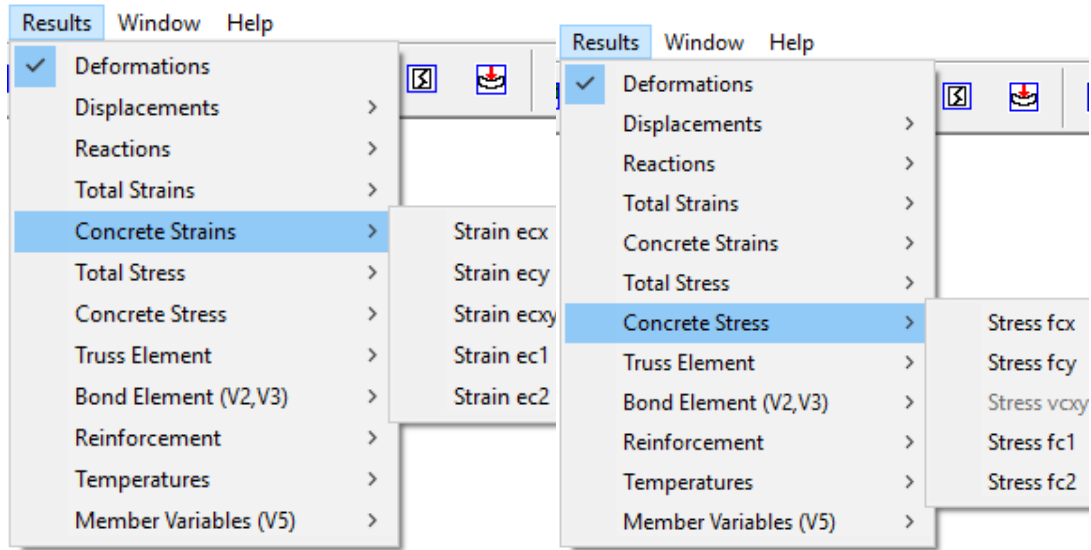

Figure 30: Results menu with highlighted parameters (a) Concrete Strains (b) Concrete Stress
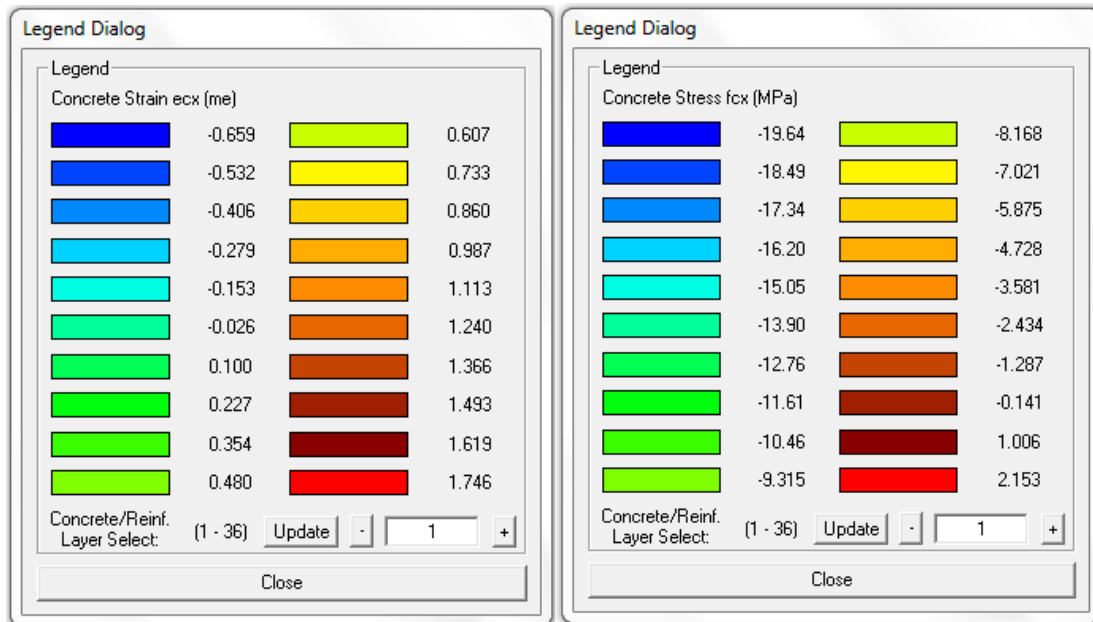

Figure 31: Legend showing points of high and low concentration (a) Concrete Strain (b) Concrete Stress
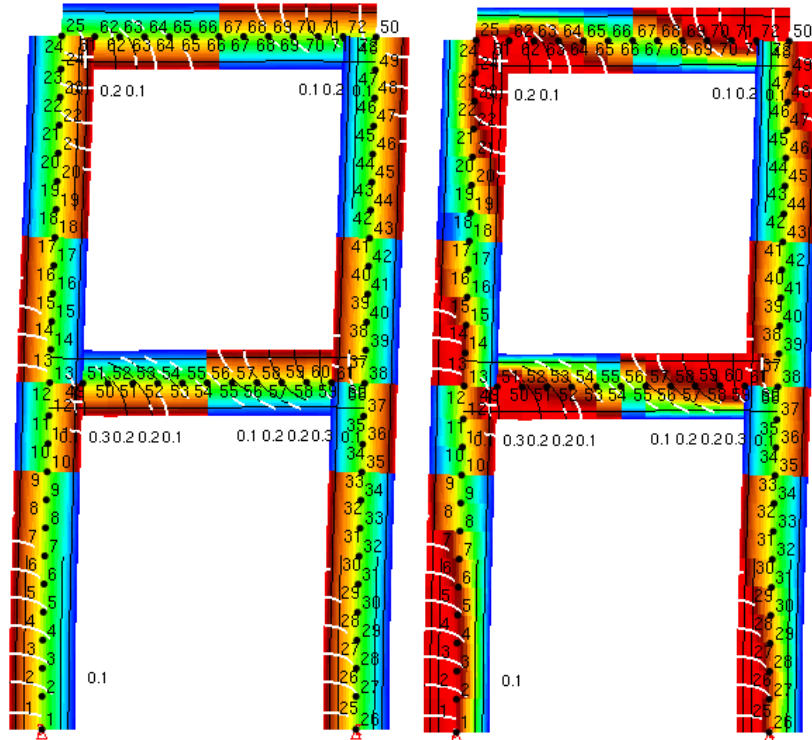
Figure 32: Results menu with highlighted parameters (a) Concrete Strains (b) Concrete Stress

7. STRUCTURE MENU

It is recommended to use structure menu which is shown in Figure 33. Load cases option gives detail about joint forces, joint displacements, moments and rotations of the frame. Restraint button provides options to select any size of restraints such as small, medium, large or even none. Material option shows different materials used along the frame indicating them with various colors. Thickness of nodes could be altered using wireframe option.
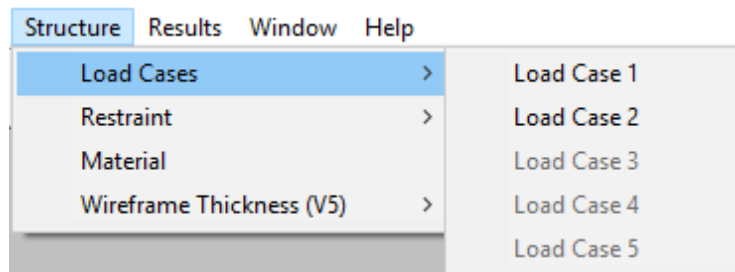

Figure 30: Results menu with highlighted parameters (a) Concrete Strains (b) Concrete Stress

This document was prepared by Agneya Sunil Loya and Daniela Dos Santos Lourenço, and edited by Salim Khoso, as a part of a project supervised by Dr. Serhan Guner.