CREATING AND RUNNING A SIMPLE MODEL

The object of this tutorial is to introduce new users to how FS200 is structured in terms of its basic operation. This will be achieved by creating and running a very simple model.

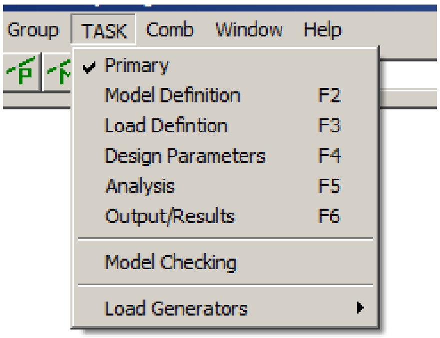
Remember! Use the Help File and press the F1 key to get Help specific to that topic.

BASIC STAGES OF ANALYSIS

The basic stages of any analysis are:

- Model Definition
- Load Definition
- Analysis - Solution of Load Cases
- Post- Processing
- Results Output

In the FS2000 GUI the basic stages are called TASKS and a menu is used for TASK selection.



When a TASK is selected, menu commands on the RHS of the TASK menu become available which are dependent upon the specific task.

OPEN A NEW MODEL

The starting point is to create the model workspace i.e. create the model data files.

From the **File** menu select **New**, the file Save-As will become visible.

Enter select or create a new folder and provide a suitable filename for the model e.g. Demo1 and click Save.

Enter model description etc. in the Save Model – Description form and click **Save.**

The Model now exists.

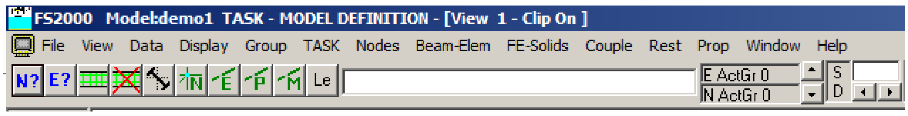
.

MODEL DEFINITION

1. BASIC MODEL DEFINITION

From the **TASK** menu, click **Model Definition**.

You are now in the Model Definition TASK. In this TASK the basic model geometry is defined.
The Menu Options to the RHS of the TASK menu will have changed and now show menus available in this TASK.



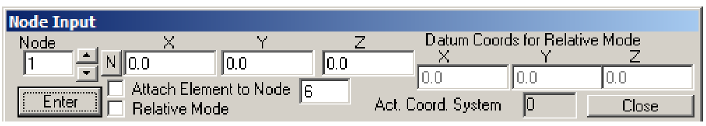
In this simple example the following definition sequence will be followed:

- Nodes
- Elements
- Element Properties
- Restraints

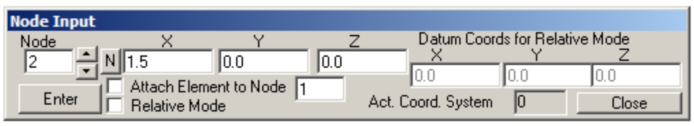
It can be seen that the sequence broadly follows the TASK menu.

2. NODE DEFINITION

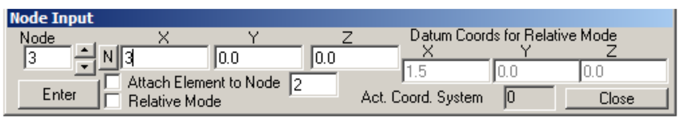
From the **Nodes** menu, click **Input**. This Node Input form will be used to define the nodes.
Click the F1 button to see the Help topic on this form (when in focus).



Click the **Enter** button to define Node 1 at the origin.



Enter 1.5 in the X co-ordinate box to define Node 2 - click the **Enter** button.



Enter 3 in the X co-ordinate box to define Node 3 - click the **Enter** button.

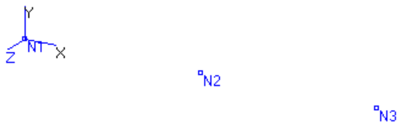
Click the **Iso View**  button (Lower LHS of Screen) to Auto scale the model.

Close the Node Input Box.

From the top button bar click the green **N** button to display the node labels.

3. ELEMENT DEFINITION

The model display should now show.



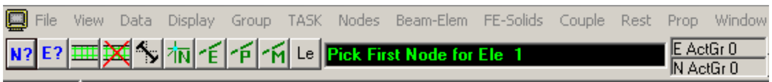
The next stage is to connect the elements to the nodes.

From the **Beam_Elem** menu, click **Input/Modify**. The Element Definition form will become visible. Click the F1 button to see the Help topic on this form (when in focus).

in the above form Node1 is the fore node of the element and Node2 is the aft node of the element.

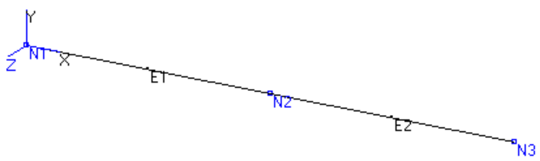
It would be possible to connect element E1 to N1 and N2 by entering the node labels in the **Node1** and **Node2** boxes and clicking the **Enter** button. This method would rarely be used, instead the nodes are picked using the mouse.

Click the **Pick Nodes** button. The Activity Status box will indicate the required user action. The ESC key or a RH mouse button click is used to cancel activities.



Use the mouse to pick (click) node N1 and then node N2. Element E1 is now defined.
Pick nodes N2 and N3 to define Element 2. Press the LH mouse (or ESC key) to end the activity.
Close the Element Definition box.
From the top button bar click the  button to display the element labels.

The model display should now show.

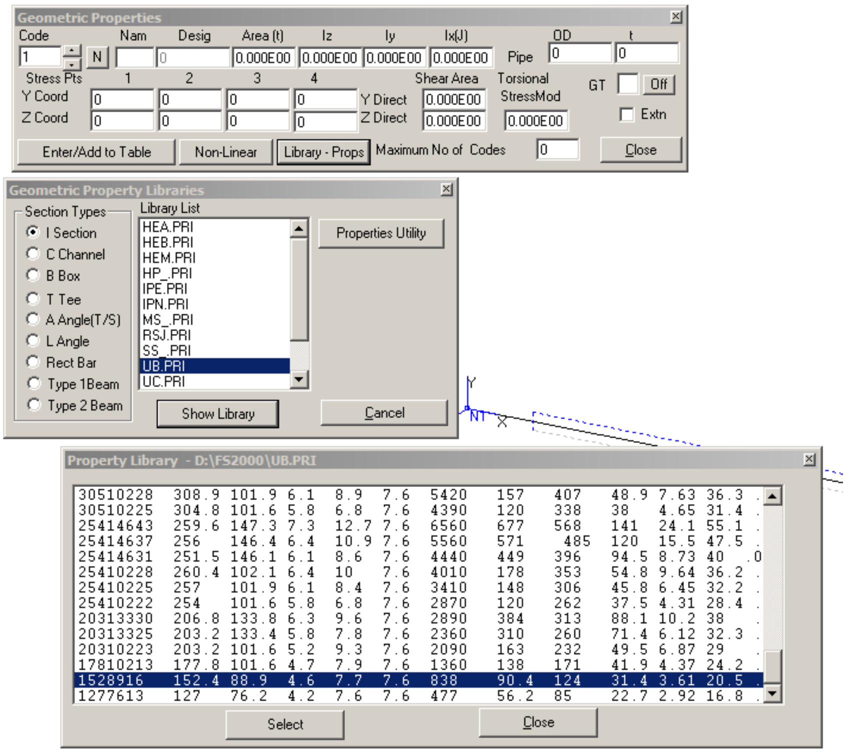


Normally there would be no requirement to display node and element labels but they are helpful in this demonstration.

The element geometric properties and material properties are defined in separate lists called property codes. The **Geom** and **Mat** boxes in the above Beam Element Definition form are used to reference these properties and thus assign them to that element.

4. ELEMENT GEOMETRIC PROPERTIES

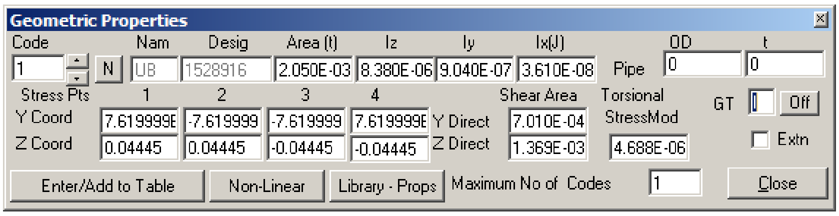
From the **Prop** menu, click **Geometric**. The Geometric Properties box becomes visible. This form is used to define the contents of the Geometric property table. In this simple model there will only be one entry.



Click the **Library-Prop** button. Select UB.PRI and click **Show Library**.

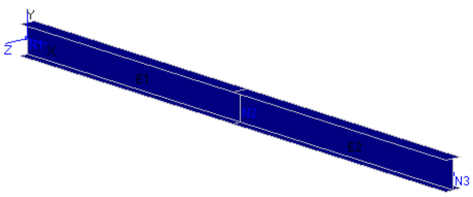
Select the 1528916 section from the Property Library.

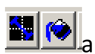
Click the **Enter/Add to Table**. The current data will be stored and the Code will be incremented to next code number in the table.
The properties are now in the table as Code 1
Use the scroll button to show the current entries for geometric property code 1



Press the F1 key to get more information on this form and its entries.

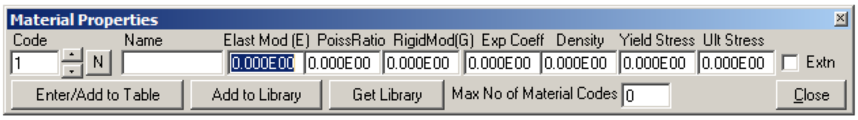
Close the Geometric Properties box.



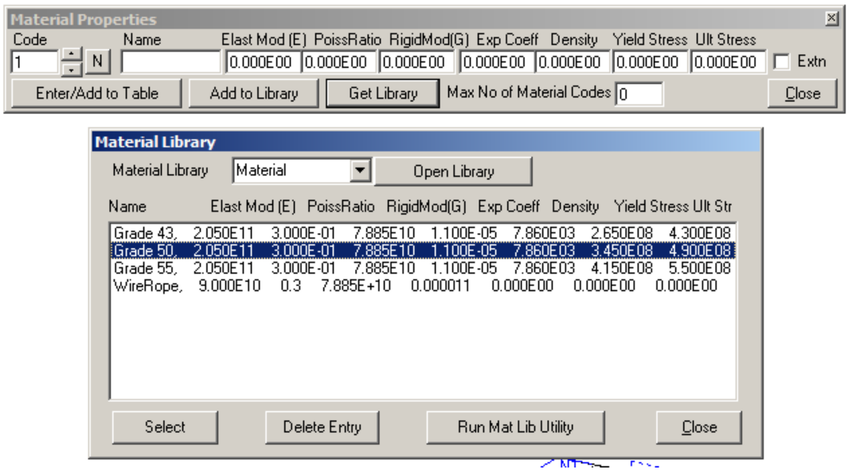
If the Virtual View and Colour Fill button  are clicked the element display should show a scaled view on the beam being modelled.

5. ELEMENT MATERIAL PROPERTIES

From the **Prop** menu, click **Material**. The Material Property box becomes visible.

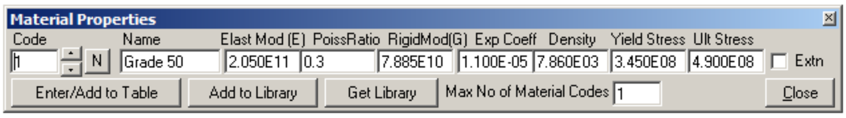


This input form is used to define the contents of the Material property table. In this simple model there will only be one entry.



Click the **Get Library** button and select Grade 50 steel.

Click the **Enter/Add to Table**. The property is now in the table as Code 1(entry 1).
Use the scroll button to show the current entries for material property code 1



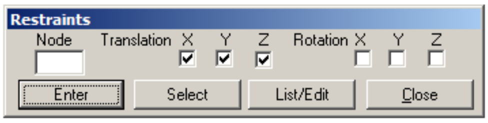
Press the F1 key to get more information on this form and its entries.

Close the Material Properties box.

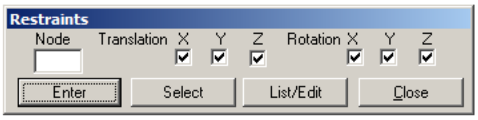
6. MODEL RESTRAINTS

The geometry and stiffness properties are now defined. It now remains to define how the model is restrained. In this example simple restraints are to be used to restrain the model. Restraints provide node fixity in the 6 degrees of freedom of the global exit system and are the easiest type of model restraint to apply, but not the only method.

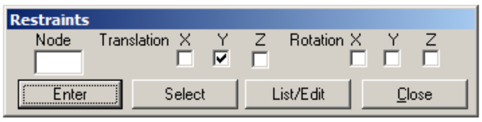
From the **Rest** menu, click **Define/Edit**. The Restraint form becomes visible.



It would be possible to apply restraints by entering the node label in the **Node** box and clicking the **Enter** button. This method would rarely be used, instead the nodes are picked using the mouse.



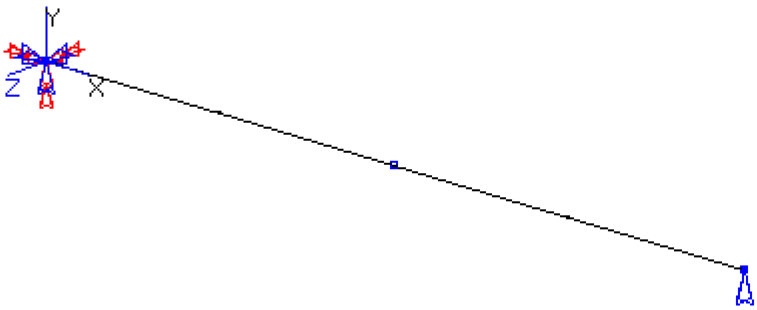
Tick X, Y & Z Rotation boxes then click the **Select** button.
Click node N1 to apply the restraints (fully fixed) to Node 1. When the node is clicked the restraints will be drawn at the node.



Remove the ticks from all restraint boxes except Y translation.
Now click node N3 to apply a vertical support to Node 3.

Close the Restraints form

The model is now restrained and the model display should now display.



The model definition Task is now complete. A propped cantilever model has been created.

The model must now be saved.

From the **File** menu click **Save**.

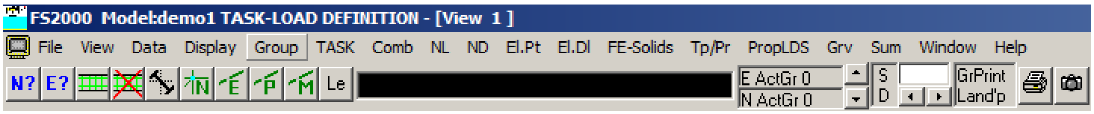
Click the **Save** button.

LOAD DEFINITION

Loads will now be applied to the model. Three load cases will be created.

- Load Case 1 - A vertical 8 kN/m distributed load + beam self-weight (gravity)
- Load Case 2 - A 10 kN vertical concentrated load at mid span.
- Load Case 3 - A 50 kN horizontal concentrated load at the tip.
-

From the **TASK** menu click the **Load Definition** command.



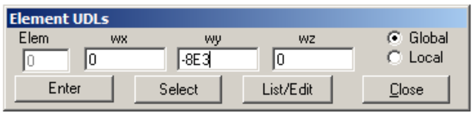
The Menu Options to the RHS of the TASK menu will have changed and now show menus available in this TASK.

When the load forms are in focus, press the F1 key to see the Help topic on that form.

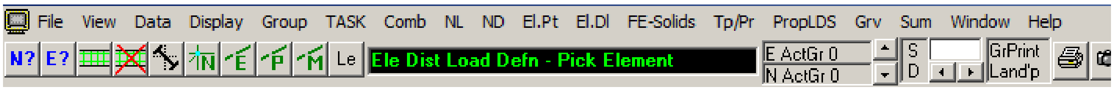
1. LOAD CASE 1

First a UDL of 8kN/m will be applied to the span. When defining loads always use fundamental units i.e. loads are in Newtons.

From the **El.Dl** menu, click the **Input/Edit-UDL** command. The Element UDL form will become visible.

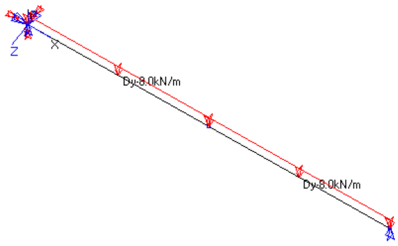


In the **wy** box enter -8e3. Click the **Select** button. The Activity Status box will indicate the required user action.



Mouse pick the centre of both elements to apply the UDL. (to pick an element it is necessary to click the centre of the element – at the yellow dot.). The red lines show that the load is applied and its direction will be indicated. Press the LH mouse (or ESC key) to end the activity.

Close the **Element UDL** form.

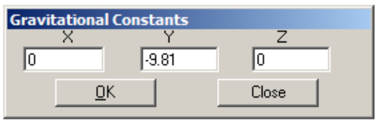


If the 

button is pressed the load magnitudes will show in the display.

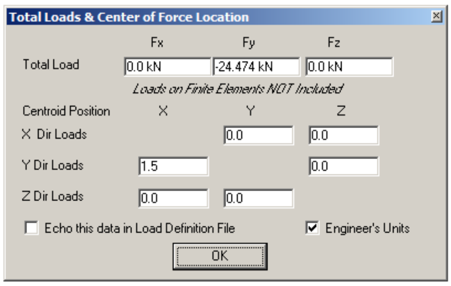
Beam self-weight is now to be applied. This is achieved by defining a gravitational constant of 9.81m/s in the –ve y direction. The loading due to the gravitational effect will be evaluated based on the density of the elements (density is defined in the Material Property data).

Click the **Grv** menu. The Gravitational Constants form will become visible.



Enter -9.81 in the Y box and click the **OK** button.

If the **Sum** command is activated the total loadings on the VISIBLE nodes and elements will be shown.



This is very useful for checking loading on the whole model or selected sections of the model. The centre of force is also listed. In this example the centre of force can be seen to be the centre of the span.

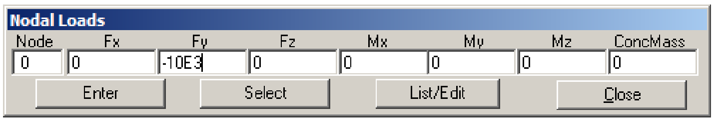
SAVING A LOAD CASE

From the **File** menu click the **Save Loads** command.
Enter 1 in the No box and a suitable description on the Description box e.g. UDL 8kN/m & Self Weigh.
A data form showing the total loading in the load case will appear.
Click **OK** to close the form. Load Case 1 now exists.

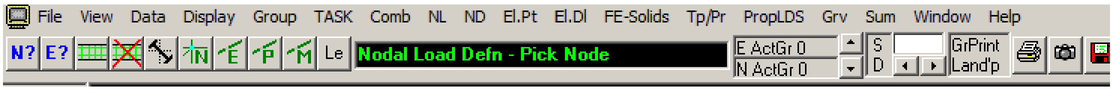
2. LOAD CASE 2

A 10 kN concentrated load will be applied at mid span.

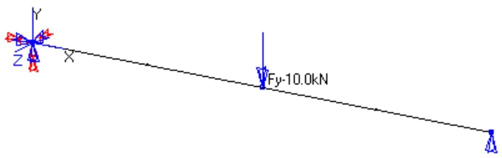
From the File menu, click New Loads. This will clear the load from the previous load case from the GUI memory.
From the **NL** menu click **Input/Edit** command. The Nodal Loads form will become visible.



Enter -10e3 in the Fy box. Click the **Select** button. The Activity Status box will indicate the required user action.



Mouse pick node N2 to apply the load to the node. The load will be shown and its direction indicated.



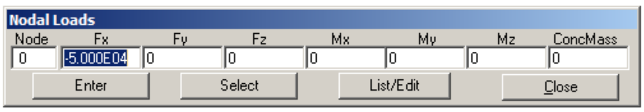
From the **File** menu click **Save Loads**, then save this loading as Load Case 2 (-10kN Vertical at mid span).

3. LOAD CASE 3

A compressive axial load of 50kN will be applied at the cantilever tip.

From the **File** menu click **New Loads**. This will clear the load from the previous load case from memory.

If the Nodal Load box is not visible, From the **NL** menu, click **Input/Edit**. The Nodal Loads box will become visible.



Enter -5e4 in the Fz box. Click the **Select** button and then click node N3 to apply the load to the end of the beam. The load will be shown and its direction indicated.

From the **File** menu click **Save Loads**, then save this loading as Load Case 3 (-1kN Horizontal at tip).

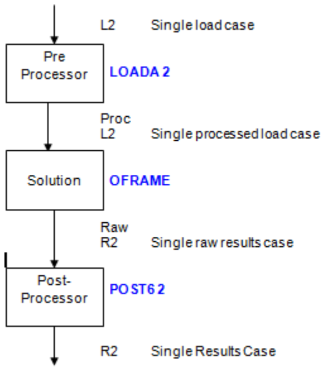
Three load cases have now been created.

In this TASK the scroll control 

can be used to scroll through the load cases.

ANALYSIS – SOLUTION

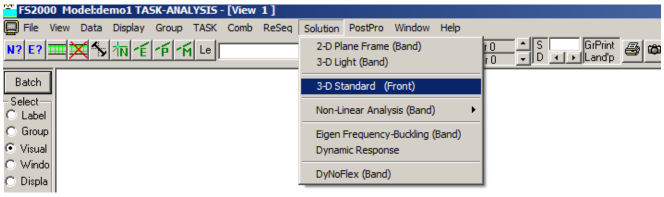
The Load Cases will now be solved. The Analysis TASK is where the different solution options are activated. In this model the 3-D Standard solver will be used. This is the most commonly used solution option for 3-D analysis. The analysis process is a 3 stage process.



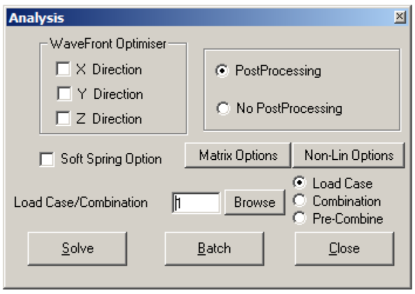
The first two stages are always linked i.e. Pre-processing must always precede the solution process. The post-processing can be done at any stage because it processes existing raw result cases from the solver.

In this example the 3 stages will effectively take place in one stage but it is worth being aware of the underlining processes.

From the **TASK** menu, click the **Analysis** command. From the **Solution** menu click **3-D Standard**.



The Analysis form will now be visible.



Click the **Browse** button and select Load Case 1.

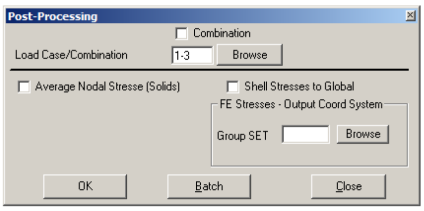
Click the **Solve** button.

On completion the Result Case 1 will be created

Repeat this for Load Cases 2 and 3 to create Result Cases 2 & 3.

The following is just a bit more background information **POST-PROCESSING.**

The **PostPro** menu command will make the Post-Processing form visible



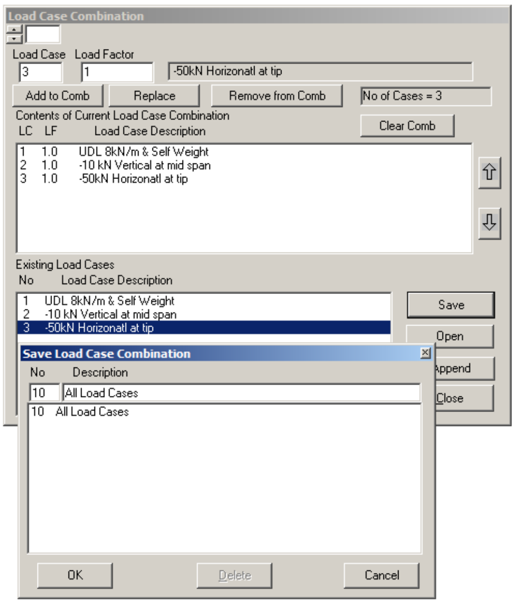
In the above Analysis form the **Post-Processing** option is active. If this were not active the raw results from the solver would have to be submitted for post processing separately. Entering 1-3 in the **LoadCase/Combination** box will post-process raw result cases 1 to 3 to create useable result cases.

POST PROCESSING – LOAD CASE COMBINATIONS

More often than not load cases are required to be combined and factored. When undertaking linear analysis the most convenient approach to this is to combine the result cases after solution i.e. during post processing. To do this a Load Case Combination has to be created.

In this example all 3 load cases are to be combined, each with a unity load factor.

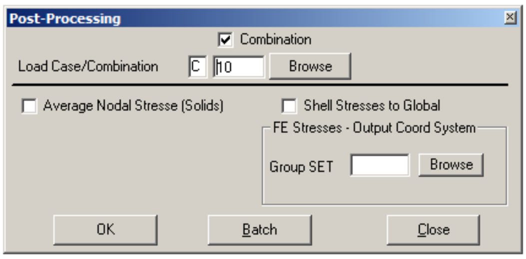
From the **Comb** menu (visible in all TASK menus) select the **Create Case Comb** command. This makes the Load Case Combination from visible. (F1 key to get Help).



Select an existing load case from the lower list box and click the **Add to Comb** to add it to the combination list box (double clicking the selection will add it to the list).

When the combination is complete ie all load cases have be added hit the **Save** button and save the combination as Load Combination 10 (LC10) using the description All Load Cases.

To combine the results use the **PostPro** menu command to make the Post-Processing form visible. Activate the **Combination** option then Browse to select the combination or enter 10 to specify C10 as the combination to be processed.



When the **OK** button is clicked C10 will be processed and Result Case 10 will be created i.e. a case in which the results from the 3 cases in the combination are summed.

INSPECTING/CREATING RESULTS

The Output/Results TASK is where the result cases can be plotted, inspected and where formatted report text files are be created.

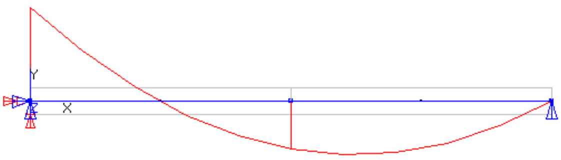
From the **TASK** menu click the **Output/Results** command.

The Menu Options to the RHS of the TASK menu will have changed and now show menus available in this TASK.

From the File menu, click Open Proc. Results and select Results Case 1.

1. PLOTTING MOMENTS ETC.

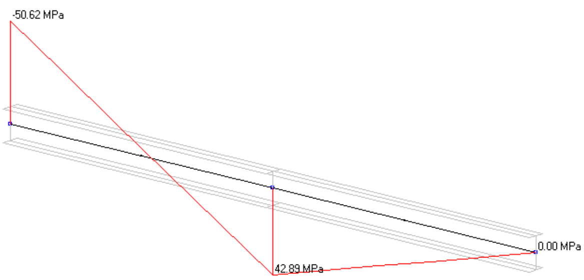
From the Plots menu, click Moment to plot the moment distribution on the beam.



The Res Toolbar is invaluable in providing quick access to controls to change the appearance of the plot. Press F1 with the Toolbar in focus

Click the button  and the magnitudes will be added to the plot.

In this TASK the scroll control  can be used to scroll through the result cases, effectively this animates the currently plotted plot.



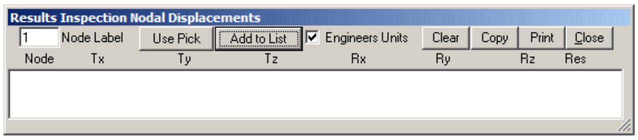
From the Plots menu select other plots to view – experiment and use the Help file.

Only nodal deflections are plotted. On a simple model like this more nodes would require to be added to produce an interesting plot.

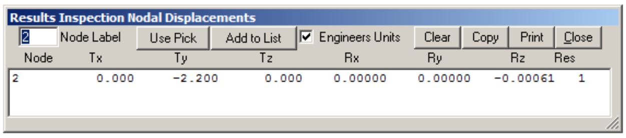
2. INSPECTING RESULTS – INTERROGATING GRAPHIC PLOTS

The commands in the **Insp** can be used to obtain more detailed result values on the displayed model.

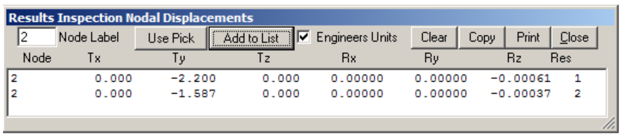
From the **File** menu open Results Case 1.
From the **Insp** menu select the Deflections command. The following form will become visible.



Click the Node Query button and then mouse pick the mid span node (N2).
Click the **Use Pick** button, this will list the displacement at N2 for the current results case.

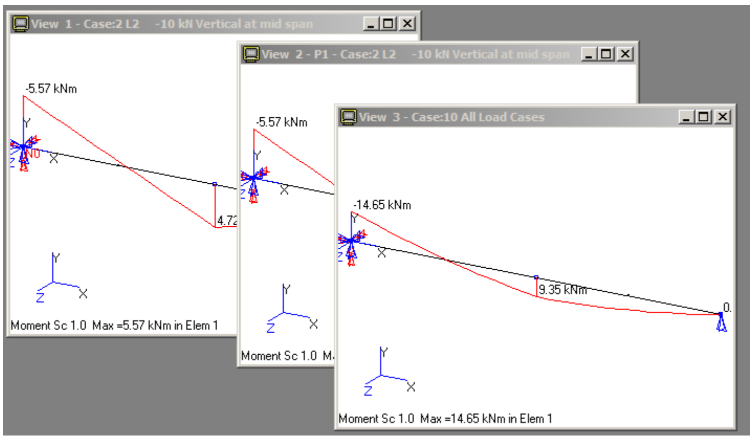


Now open Result Case 2 (or scroll to it) and click the Use Pick button again. The deflections at N2 for Result Case will be appended to the list.



Element forces & stresses and reactions can be similarly inspected and listed.

It is possible and sometimes useful to open additional Windows and open different result cases in different Windows.



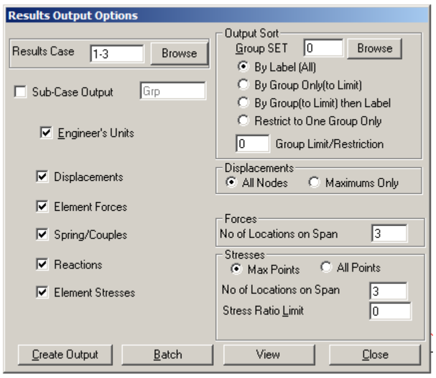
CREATING RESULTS OUTPUT – FORMATTED TEXT

Text output showing displacements, element forces etc. can be created for each result case. Formatted text output is not created by the post-processor but by additional selective processing in which the user controls the output.

1. STANDARD INDIVIDUAL RESULTS

Formatted results output for the 4 result cases will be created

From the **StdOut** menu click **Individual Results Format**.
The Results Output Options box will become visible.



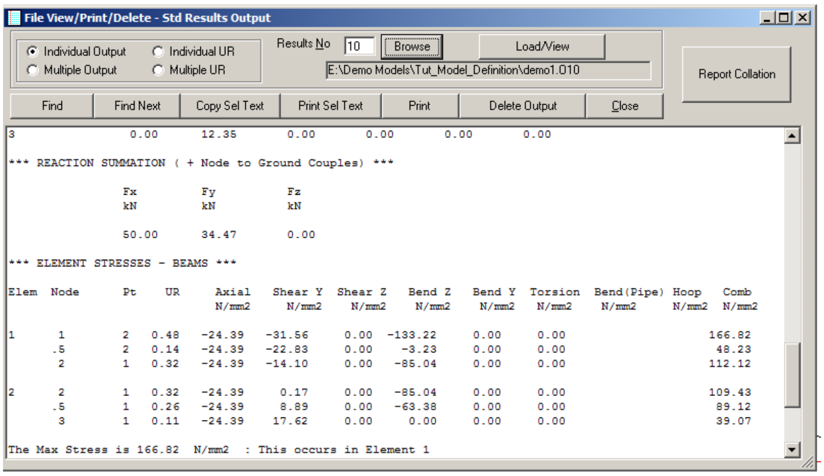
Enter 1-3 in the Result Case box
Enter 0 in the Stress Ratio Limit box (no stress limit required on such a small model)

Click the **Create Output** button to create the formatted data for case 1 to 3.

Use the **Browse** button and select Case 10 and create the output for that case.

To view the file, click the **View** button. The Output Viewer will become visible.
Click the Load/View File button to load Results Case 10.

The formatted output file for Results Case 2 will now be visible. It can be viewed or printed etc. Note that the Print setting form (File menu :Print Setting command) can be used to set the font etc.



The results may also be viewed from the **Data** menu using the **View/Print Data** command (Standard Analysis Results sub-menu). Try viewing the file from the **Data** menu.

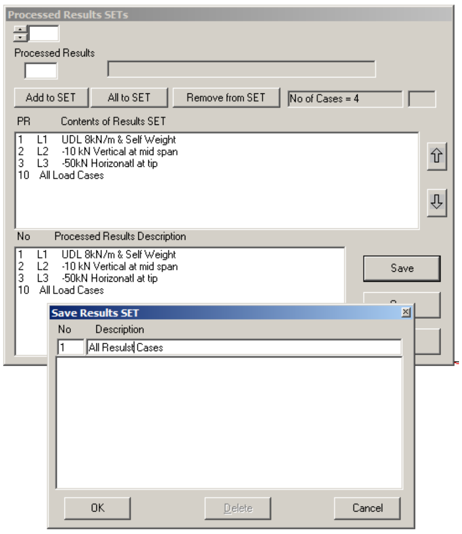
2. MULTIPLE RESULTS

Sometime it is convenient to create output in which the results for different cases are listed by entity grouping e.g. element forces by element label. A result SET is used to identify the cases to be included.

RESULTS SETS

To do this requires the use of Result SETS. A Results SET is simply a combination of processed Results Cases. In this example a Result SET which contains all the cases will be created.

From the **Comb** menu (visible in all TASK menus) select the **Create Result SET** command. This makes the Load Case Combination form visible. (F1 key to get Help).



Click the **All to SET** and add all the results to the SET list box.

Click the **Save** button and save the combination as Result SET 1 using the description All Result Cases.

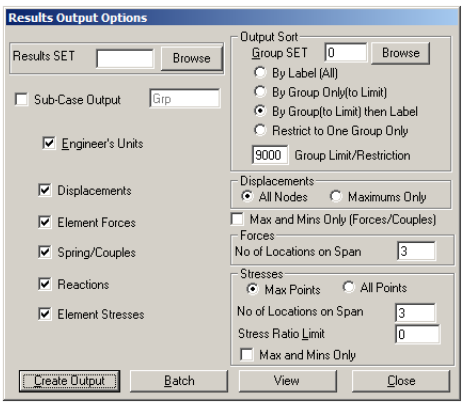
This SET will now be used to create a multiple results formatted output.

Repeat the process to create SET 2 but only cases 1 to 3 will be included.

We now have 2 Result SETS

CREATE THE MULTIPLE OUTPUTS

From the **StdOut** menu click **Multiple Results Format**.
The Results Output Options box will become visible.



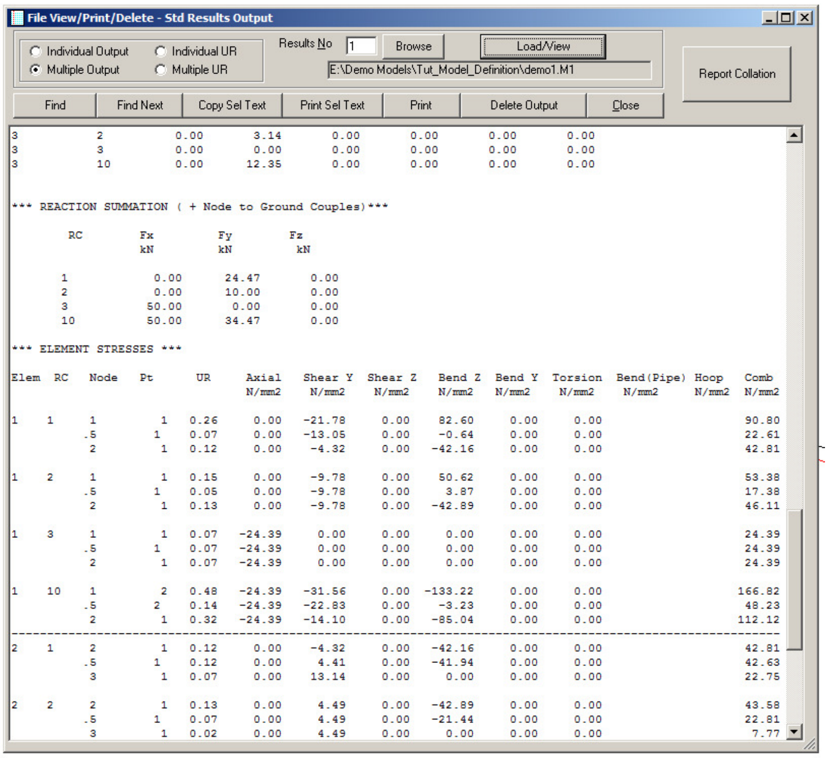
Enter 1 the Result SET box
Enter 0 in the Stress Ratio Limit box (no stress limit required on such a small model)

Click the **Create Output** button to create the formatted data for case 1 to 3.

Use the **Browse** button and select Case 10 and create the output for that case.

To view the file, click the **View** button. The Output Viewer will become visible.

Click the Load/View File button to load Result SET 1.



The output file is paginated only when the file is printed.

3. UR – UTILISATION RATIOS (PLOTS AND LISTS)

In this model with few elements and few load cases it is very easy to see how the model performs. In a large model with a large number of result cases it is sometimes difficult to visualise element performance.

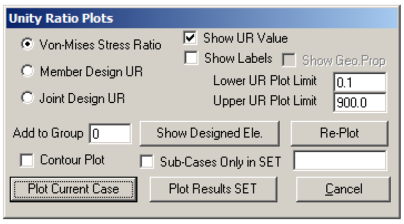
UR plots and lists are extremely useful in showing the performance of individual elements either from a single result case or from multiple result cases in one plot or one list.

In this example the UR values will be based on the Von-Mises stress ratio which is obtained when a result case is processed using the Standard Individual Results option. UR values are also obtained from the various optional design code checkers e.g. AISC member design.

UR PLOTS

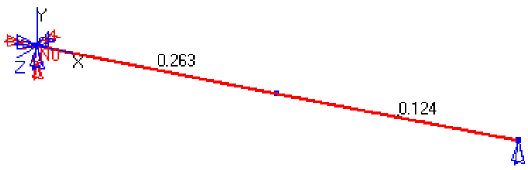
Click the UR button 

or the UR (Unity Ratio) command from the Plot menu. The Unity Ratio Plots form becomes visible.



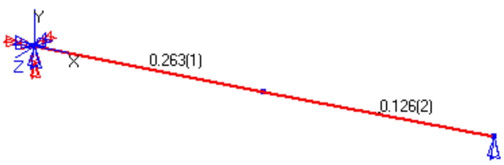
Click the **Plot Current Case** button.

If case 1 is current the following will be displayed.



Use the result case scroll and see how the UR values change.

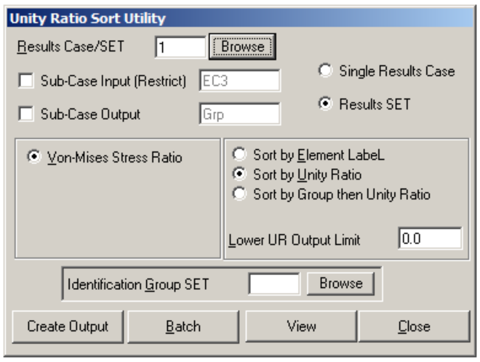
Click the UR plot button and now this time click the **Plot Results SET** button and select SET 2. The following will be displayed.



What the UR plot shows is the maximum UR for each element for all cases in the SET. The value in the ( ) is the case number that produces the maximum.

UR LISTS

A formatted list equivalent of this plot can be obtained using the URSort utility.
From the **StdOut** menu click **Sorted Unity Ratios** command. This will make the following form visible.



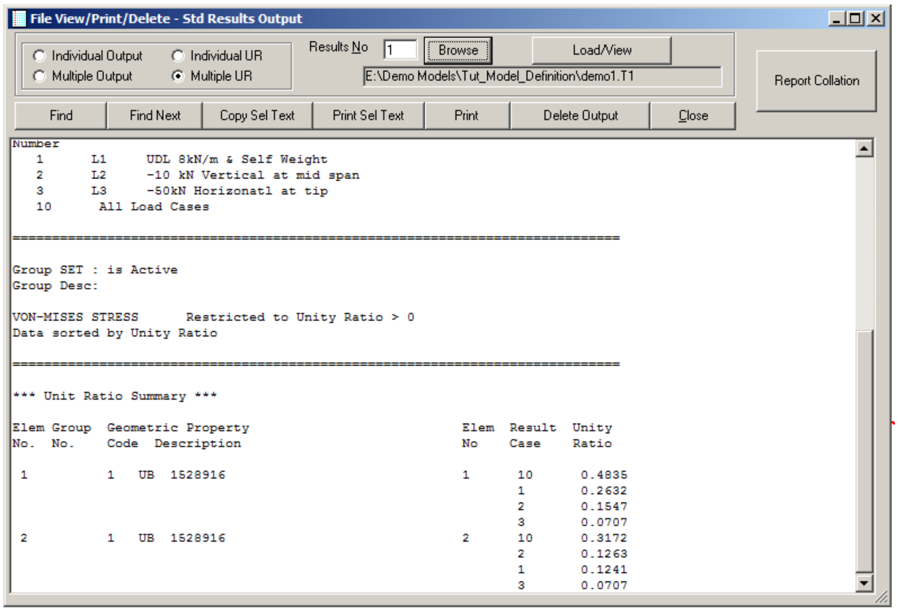
Enter or Browse for Results SET 1

Set the **Lower UR Output Stress** to 0

Click the **Create Output Button**

Click the **View** button.

Click the Load/View button to load the processes UR output.

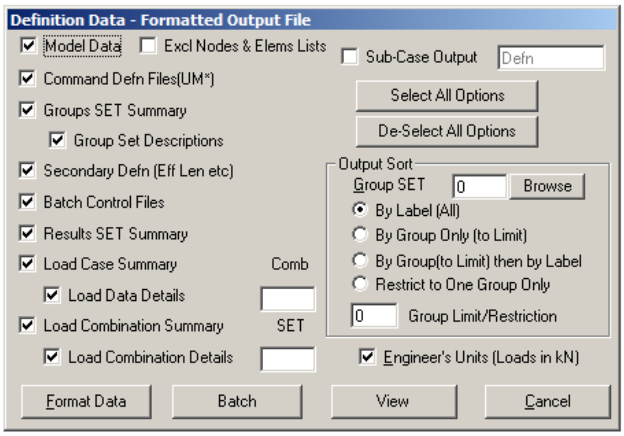


This list shows the highest Von-Mises utilisation in the model, listed by element label. For each element, the UR for up to the 10th highest cases will be listed in ascending order.

FORMATTING DEFINITION DATA

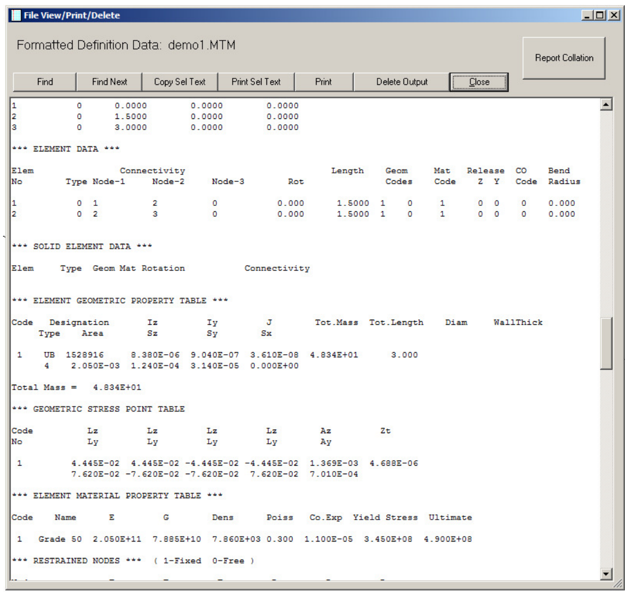
The model data, both model and load definition data, may be processed to produce formatted text data which clearly lists all definition aspects of the model.

From the **Data** menu, click **Format Definition Data**. The definition data box will become visible.



In this example all model data will be **formatted**.

Click the **Format Data** button. When processing has stopped click the **View** button.
The Output Viewer will display the input data in Report Format.



This output may also be viewed from the Data menu using the View/Print Data command (Formatted Definition Data sub-menu). Try viewing the file from this route.

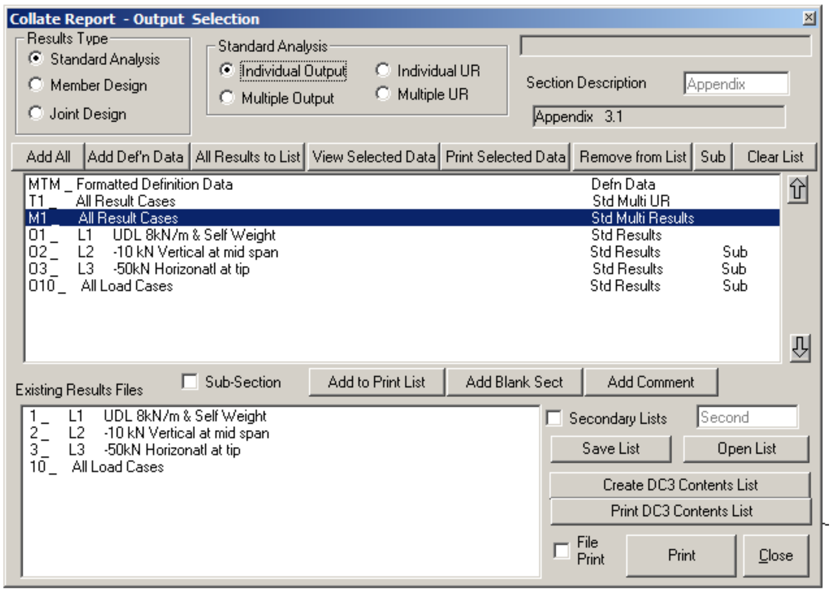
The formatted data will be paginated when the file is printed.

CREATING A REPORT

When working with a model a large number of formatted output files will be created. These output files include both definition data and results data.

It is nearly always a requirement to compile these separate files into a structured report. A collation utility for doing this is provided in FS2000.

From the **Data** menu click the **Report Collation - Data Selection** command.
The Collate Report - Output Selection form will become visible.



If the Add All button is clicked all formatted data produce will be added to the print list i.e. the upper list box.

Selected files in the print list can be viewed by clicking the **View Selected Data** button.

If a PDF writer is available the whole of the model data can be written to a PDF file in one go.

It is not the intention here to describe the use of this report/print utility, only to show its availability. The Help file describes its operation in more detail (F1 key).