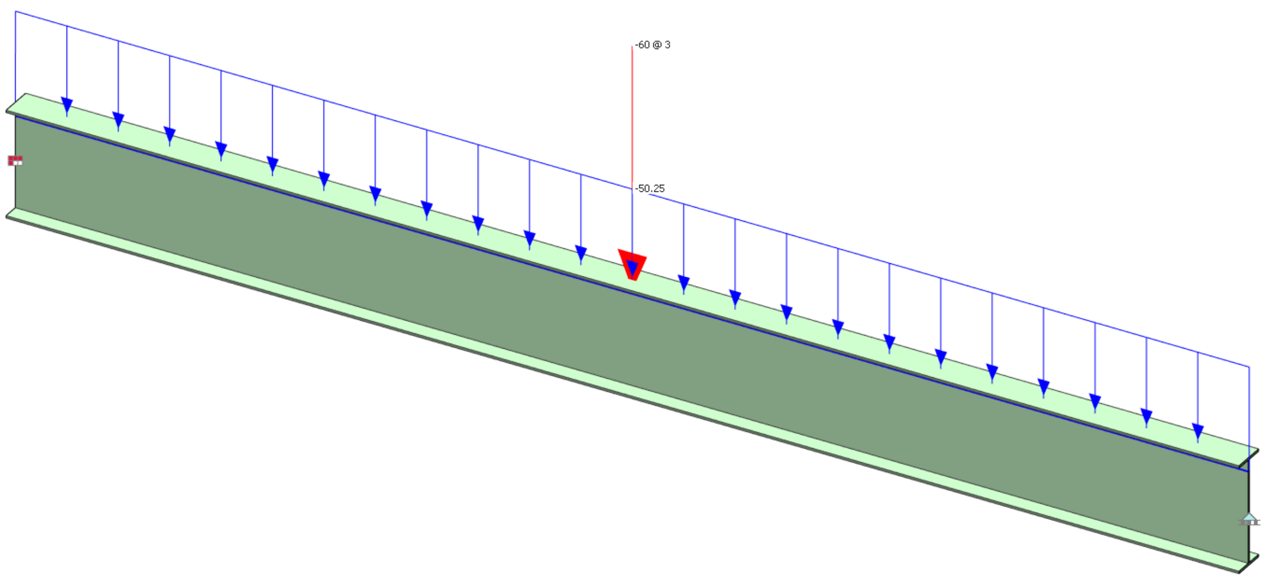
masterSERIES

T01-1 MasterFrame Tutorial

Simple Steel Beam Design to Eurocode 3



Contents

Introduction

This tutorial will require you to have access to MasterFrame and MasterKey: Steel Section Design or PowerPad.

> *MasterFrame is a piece of advanced structural frame analysis software that allows you to analyse everything to beams, and trusses to multi-storey frames and complex 3D models in any type of material.*
>
> *MasterKey Steel Design allows for the design of members in steel structures analysed using MasterFrame, or MasterPort to the BS, Euro or SABS codes.*

If you do not have access to any of this software, please contact us for a 14-day free trial to learn how it can benefit you and your business.

Overview & Outcome

In this tutorial, we will undertake the development and design of a 3-bay portal frame with the aim of providing you with a solid understanding of:

- Defining the span, properties and loadings
- Setting the support conditions
- Analysis
- Design of steel beam
- Viewing and printing the results

Version Information

This tutorial has been written for version 2020.01 of the MasterSeries software suite. Subsequent versions of the software may have additional features or changes in layout, however the general procedure will remain the same.

Contact

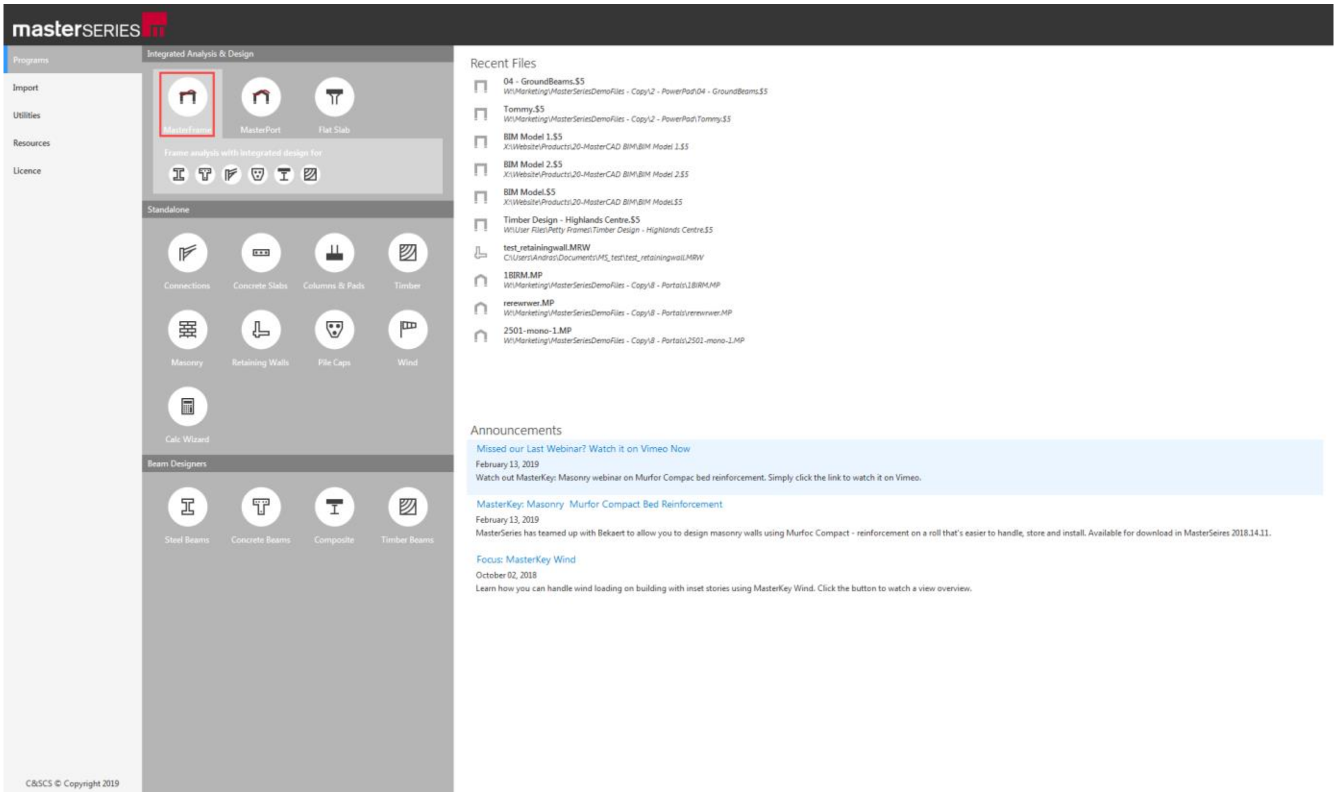
We strive to make our tutorials as simple as possible without compromising on the technical aspects of the analysis procedure. Should you discover any errors, omissions, or are in need of additional clarification, please contact us by emailing your comments, or corrections to help@masterseries.com.

We're social – follow us on Facebook, LinkedIn and Vimeo to keep up to date!

Loading MasterFrame

To start this tutorial, launch the main program of **MasterSeries**.

While standing on the **Programs** tab, select the **MasterFrame** from the *Integrated Analysis & Design* filed.

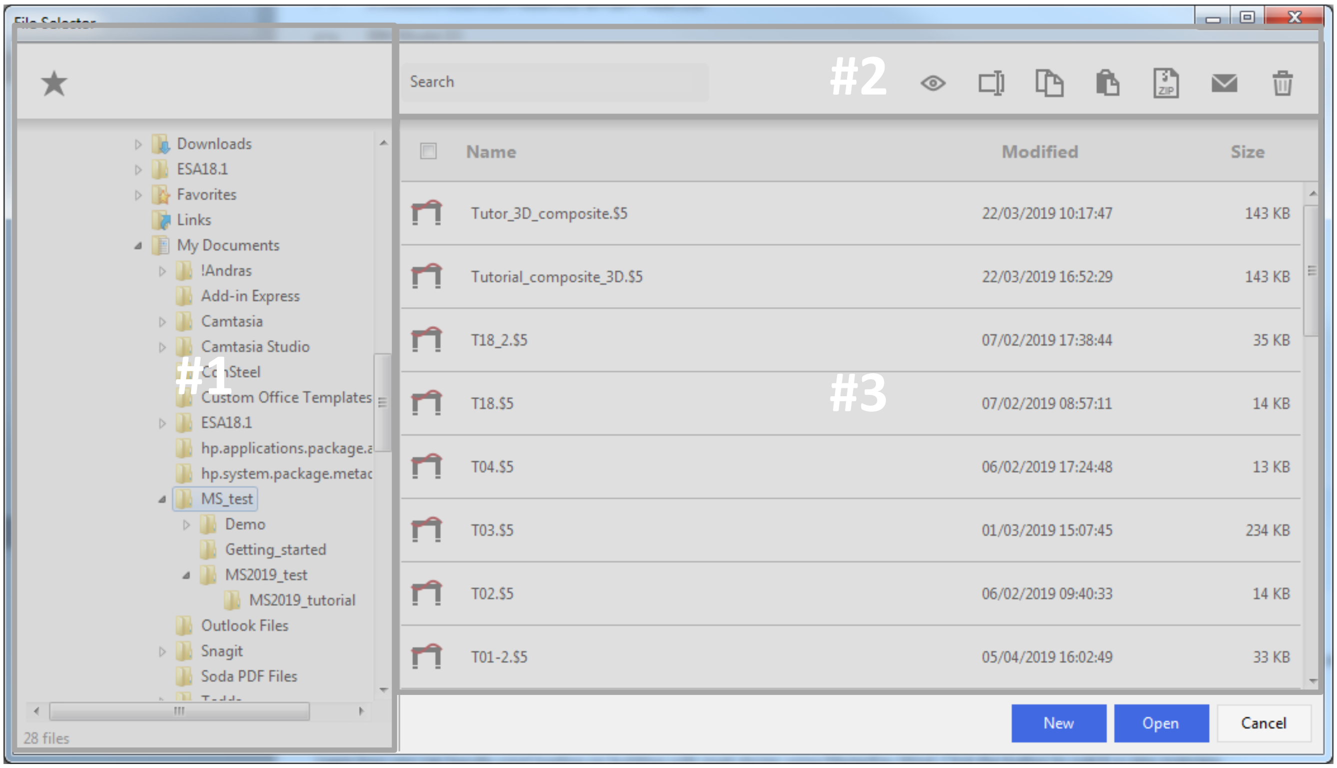


Hovering over the **MasterFrame** icon, the available integrated design options appear with small icons.

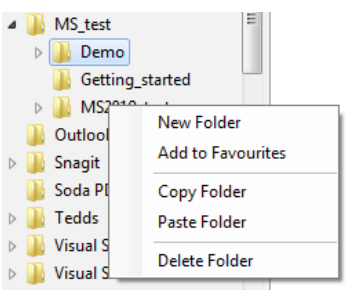
The File Selector dialogue

The **File Selector** dialogue will now be displayed.

You can use the **File Selector** to navigate in your folder tree and to select, modify or delete your existing model files or create a new one.



The left side of the **File Selector** is a usual Windows file tree (#1) which can be used to navigate between the folders. By clicking with the right mouse button on one of the folders in the tree, the following functions will be available:



1. Create a new folder in the selected folder
2. Add the selected folder to Favourites to create a shortcut
3. Copy selected folder to the clipboard
4. Paste the content of the clipboard to the selected folder
5. Delete the selected folder.

By clicking on the **Star** () button on the top of the file selector tree, we can see the saved favourite folders. By selecting one of them, the file tree will immediately jump to there. By clicking the **Star** icon at the end of the line, we can remove or add each of them.

The table, on the right of the **File Selector** (#3), lists all of the **MasterFrame** models contained in the selected folder.

Just like in any other table application, by clicking on the header, the content of the table can be ordered by name, modification date or size.

Selecting one of the model files, it can be opened by clicking on the **Open** button or using the top tool bar's functionalities (#2), it can be manipulated.

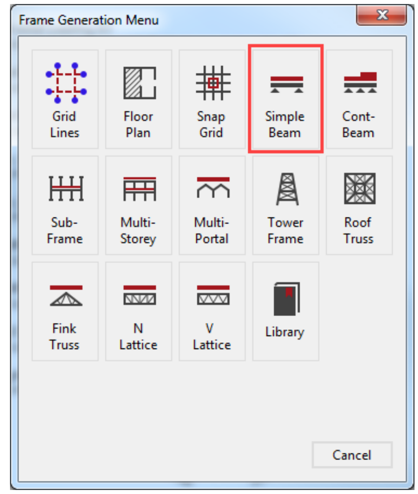
- View/hide the model preview window
- Rename the selected model file
- Copy the selected file(s) to clipboard
- Paste the selected content of the clipboard to the actual folder
- Zip the selected file(s) with all the additional files
- Zip and attache to a blank email the selected file(s) with all the additional files
- Delete the selected file(s)

Multiple file selection is available to select the files while pressing the **Ctrl** button, or using the range selection by selecting the first one then select the last one while pressing the **Shift** button. Or all of the files can be selected/deselected by clicking in/out the checkbox on the top of the first column.

To create a new model file click on the **New** button on the bottom, then type the name (for example Tutor01-1) and click on the **Create** button.

The Frame Generation Menu

In most cases, you will be able to select a start-up frame and then tailor it to your specific requirements. In this case, we shall generate a single-span beam.



The **Frame Generation Menu** (Frame Wizard) is now displayed as shown.

> **TIP!** If the frame you are generating does not match one of the pre-processor frames, choose a frame that is similar to, but larger than, your frame. It is easier to delete members than to add them. If in doubt a multi-storey frame makes a good basic grid.

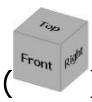
To generate our start-up frame, select the **Single Beam** button.

☞ **Chapter 3 MasterFrame - 3.3.1 Frame Generation Procedures and Templates**

This tutorial describes some of the basic techniques used in MasterFrame. Please take a few minutes to familiarise yourself with the various frame viewing tools; editing and data input methods and find how you can use the modify geometry area to select members.

Defining Member Section Properties

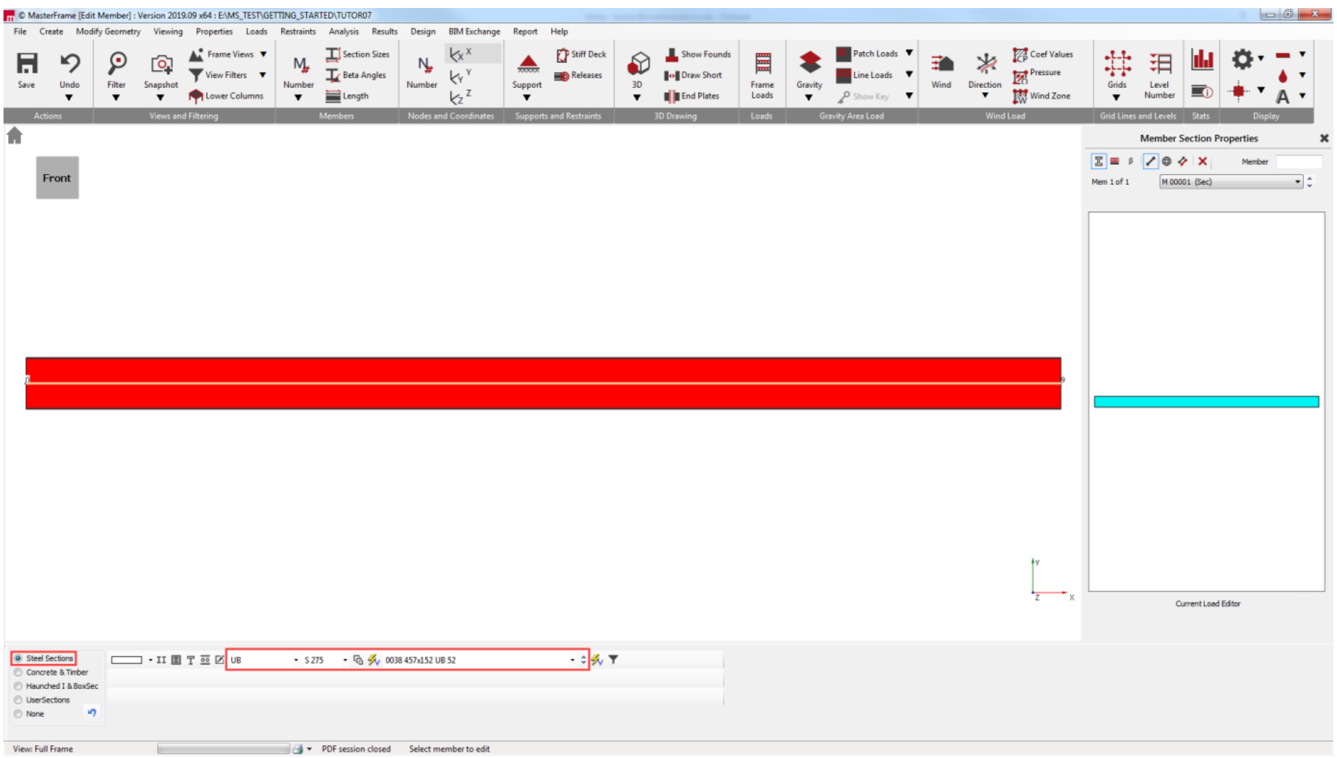
The **Member Section Properties** is now displayed. This allows you to set section properties to the beam.

Place the beam in front view by selecting the **Front** side of the viewing cube ().

Select **Steel Sections** from the bottom left-hand corner and from the section sizes droplist, select a **457x152 UB 52**, grade **S275**.

The section size will have been applied to the simple beam.

> **TIP:** Additional information about the beam can also be added, for example, any haunches applicable to the member, compound sections with plates, double members, cellular sections, or concrete encasement.



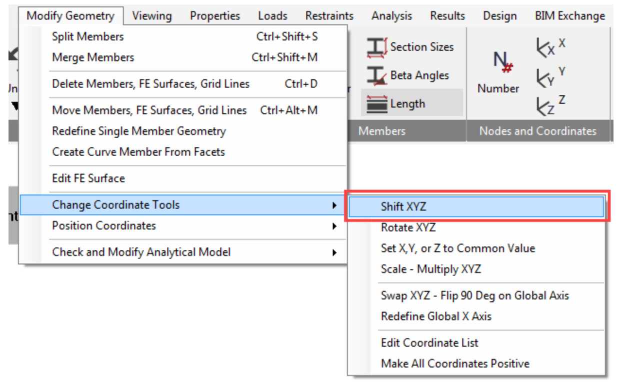
Close the **Member Section Properties** dialogue by clicking on the **X**.

Editing the Geometry

You can now use the **Modify geometry** options to modify the frame geometry.

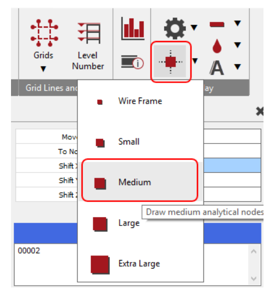
From the top toolbar turn **on** the member **Lengths** () in the **Members** group to see the actual length of our beam.

From the **Modify Geometry** menu and **Change Coordinate Tools**, select the **Shift XYZ** option.



Turn on Node Dots and set them to Medium

Select the beam's right endpoint. A red dot will appear and the node number will be listed in the **Nodes to Shift** list to confirm the selection.



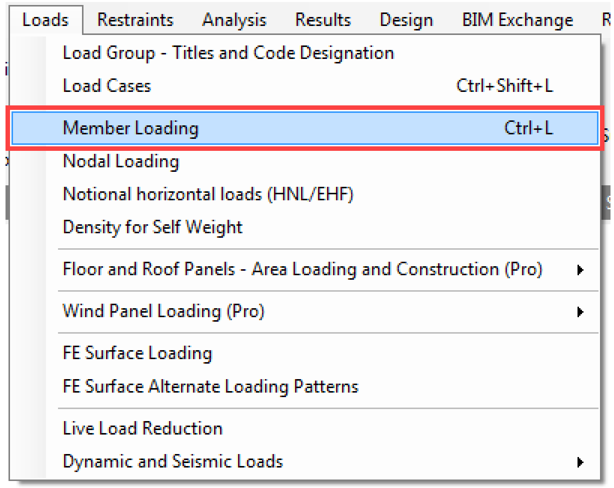
Therefore we would like to shorten the beam from 9 m to 6 m, enter **3** in the **Shift X (m)** box.

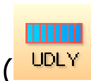
Select **Apply** to carry out the shift.

Close the **Shift XYZ** dialogue by clicking on the **X**.

Applying Member Loading

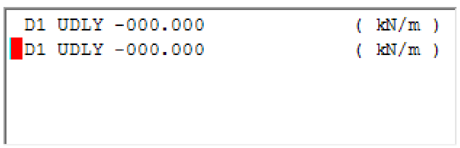
To apply loading to a member, select **Member Loading** from the **Loads** menu.



Click on the **UDLY** (

) button (bottom left of the screen) **twice** to add two uniform distributed load definitions.

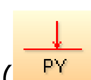
The list of loads edit box now has two loads in it. Both loads are D1 UDLY -000.000 which means both of them assigned to the D1 dead load group, both of them uniform distributed load in the Y-Axis and acting downward.



We now need to edit these loads.

Pick the upper of the two loads to set the cursor focus to it and change the load to **D1 UDLY -015.000 (kN/m)** by overtyping the **- 000.000** value.

Next, pick the second load or move your cursor along to it and change the load to **L1 UDLY -020.000 (kN/m)** by overtyping **D1** with **L1** to assign the load to the **L1** live load group and overtyping **-000.000** value with **-020.000**.

Click on the **PY** (

) button to add a point load definition.

Pick the point load in the list of loads edit box and change the load to **L1 PY -040.000 3.000 (kN, m)**.

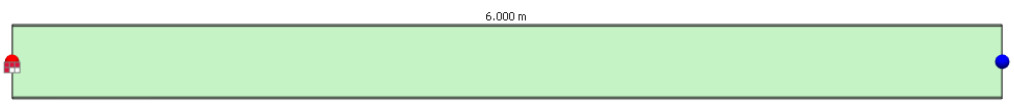
Close the **Member Loads** dialogue by clicking on the **X**.

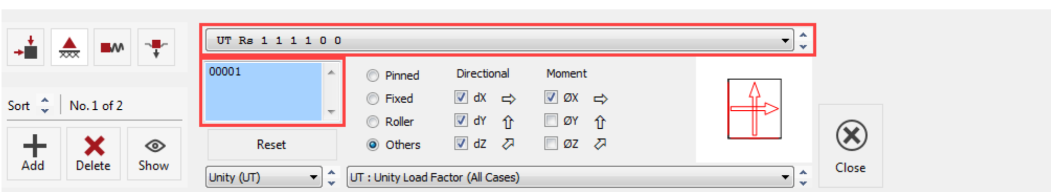
Nodal Supports

The **Single Beam** wizard already defined a **Pinned (dX, dY, dZ) with a ϕX restraint** nodal support on the left-hand endpoint of the beam (node 1) and a **y(dY) direction only restraint** on the right-hand endpoint of the beam (node 2).

In this tutorial, we will not change the support condition, but introduce the nodal information area, and how to view the support conditions.

From the **Restraints** file menu, select **Nodal Static Supports**.



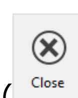


Using the dropdown list we can change between the created support conditions.

List of nodes edit box shows the number of the nodes where the selected support is applied.

Each support is graphically represented on the structural model.

> **TIP!** If you view the frame in Orthogonal view (instead of front view) by selecting the home button . Then you get a better idea of the restraints applied on each node .

Close the dialogue by clicking on the **Close** (
) button.

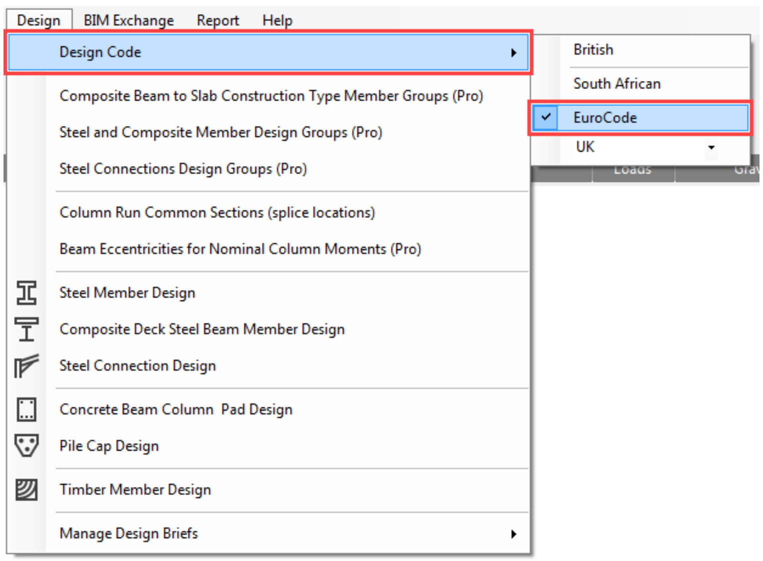
Load Cases

Design Code

We need to make sure we are designing to **Eurocode**.

On the top menu, go to **Design > Design Code** and check the selected code which should be **Eurocode**. If not, please select it.

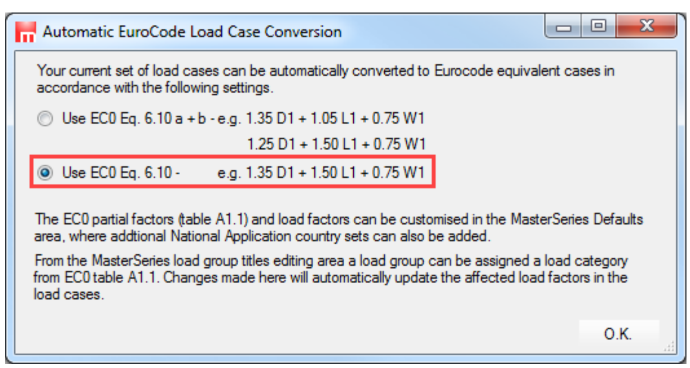
If you are asked *"Do you wish to change the Loading Cases in accordance with the design code change"* select **Yes**.



Go again to the **Design > Design Code** and check the national annex which should be the **UK**. If not, please select it.

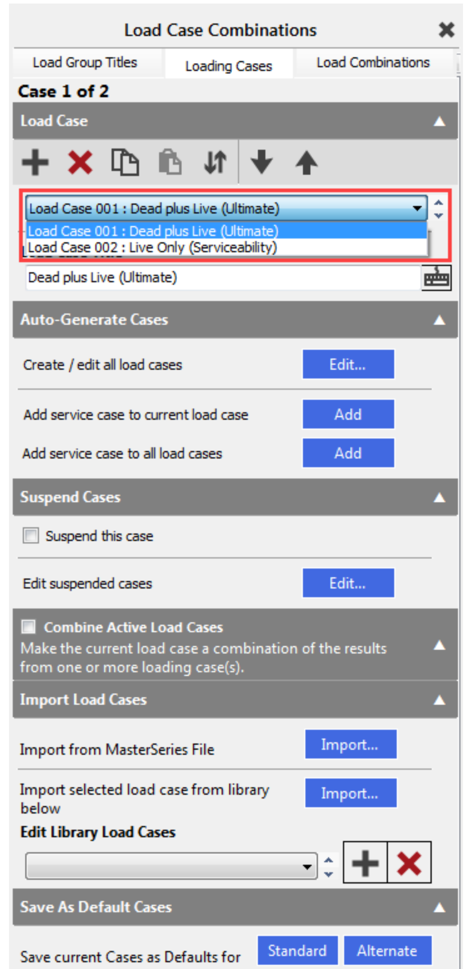
If you are asked *"Do you wish to change the Loading Cases in accordance with the design code change"* select **Yes**.

If you are asked which *EC0 Eq. 6.10* set to use, choose the lower option **6.10**.



Note: Even if you are from a different country, choose the UK so that all the loads and design parameters match those in this tutorial.

Loading Cases



From the **Loads** menu, select **Load Cases**.

By default, there are already two (or three in case of using EC0 Eq. 6.10 a+b by default) combinations of actions and case titles generated.

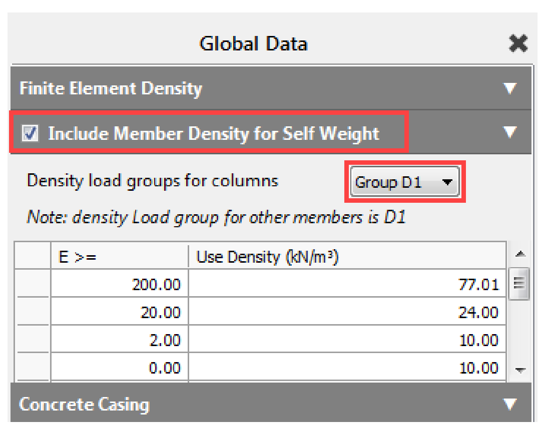
Load Case 001: **Dead plus Live (Ultimate)** (Permanent Plus Variable)

Load Case 002: **Live Only (Serviceability)** (Variable Only)

In this tab, we have the facilities to manage loading cases and define titles for them. However, for this example, we will retain the default loading cases and titles.

Density for Self Weight

Self-weight of structural members is automatically included as a default set up.

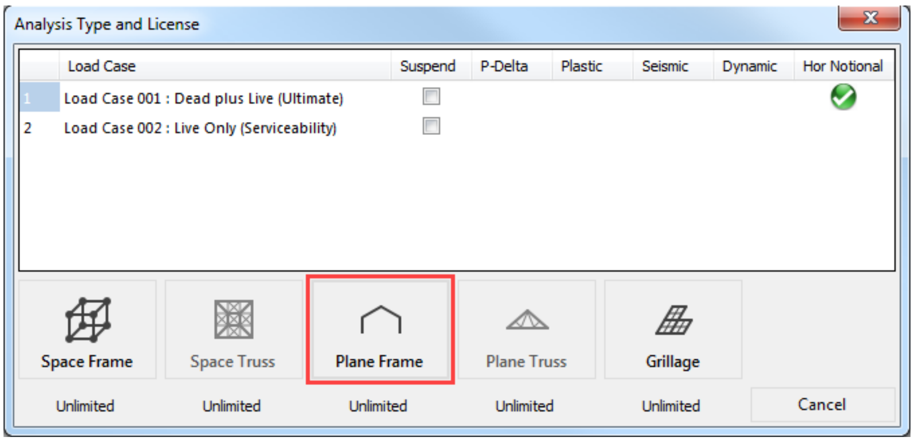


The values for self-weight can be checked by going to **Loads > Density for Self Weight**. To ensure that the self-weight is included, make sure the **Include Member Density for Self Weight** checkbox is ticked. The default load group of the self-weight is the **D1**, but using the drop-down menu we can modify it.

Analysing the beam

From the top menu select **Analysis**, then **Static Analysis**.

The file will be saved automatically and the **Analysis Type and License** toolbar will now appear.

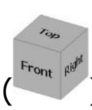


Select **Plane Frame** () and the frame will be analysed.

> **Note:** In Space/Plane Frame analysis, members are assumed rigidly connected together. The user can pin specific members by using the Member End Releases function in the Restraints menu.
>
> In Space/Plane Truss analysis, all loads on members are converted to Nodal Loads and there is no bending of the members.

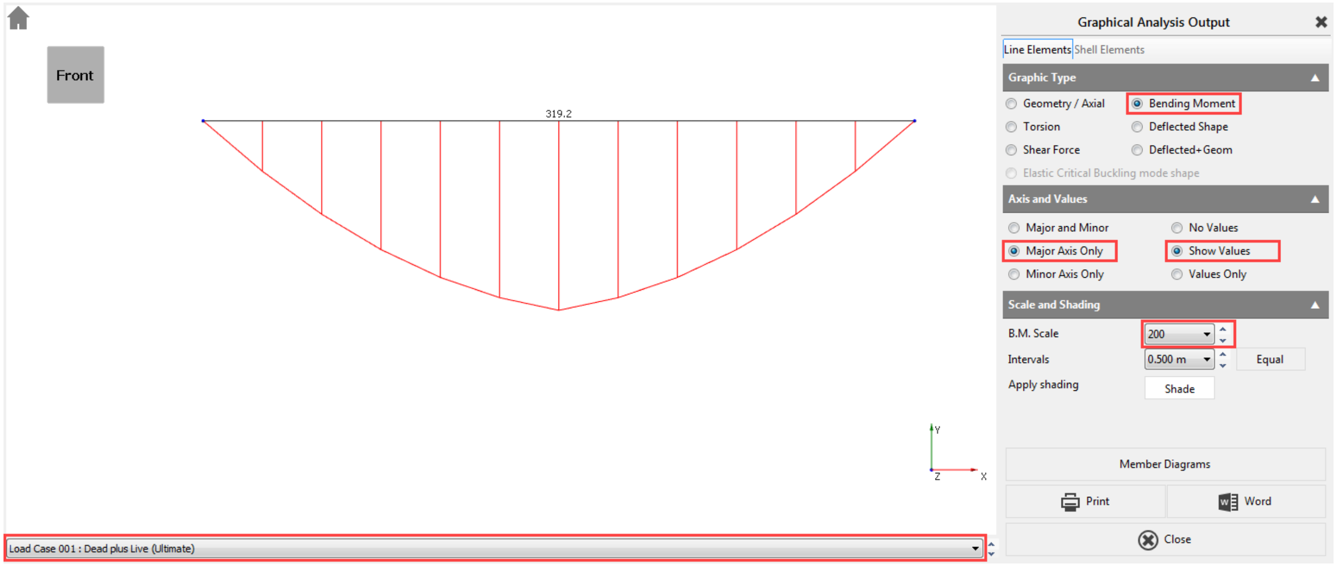
Viewing Graphic Results

From the **Results** file menu select **Graphical Analysis Results**.

Place the beam in front view by selecting the **Front** side of the viewing cube () and turn off the **3D**, if it is on.

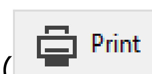
Select **Bending Moment**. We have no bending about the **Minor axis** so select **Major Axis Only** on the right-hand panel.

Select **Show Values**. Change the **B.M. scale** to **200**.



Using the drop list, at the bottom of the screen, we can select and view each of the **Load Cases** including the envelopes.

Printing Graphic Results

Select the **Print** (

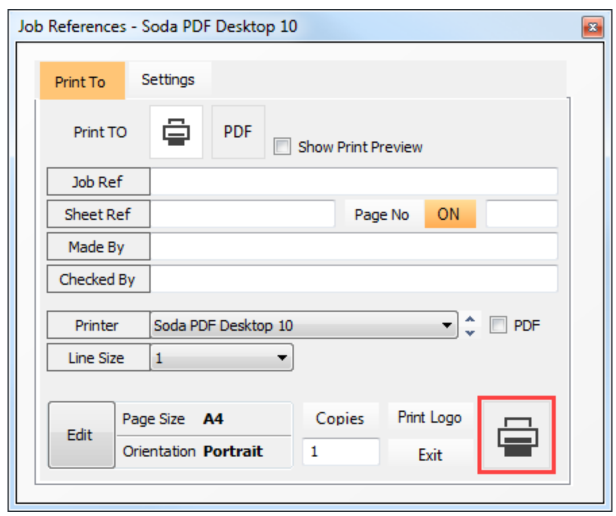
) button.

The **Job Reference** form enables you to finalise the job details on your printout just before printing them.

The **Job Reference** text boxes enable you to edit the job reference details that will appear in the printout.

While viewing the preview you can use all the tools on the top toolbar and the side menu, including zooming, panning, scaling and font sizes.

You can change the printer from the dropdown list of printers and alter the page orientation from portrait to landscape.



Select the **Printer** () button.

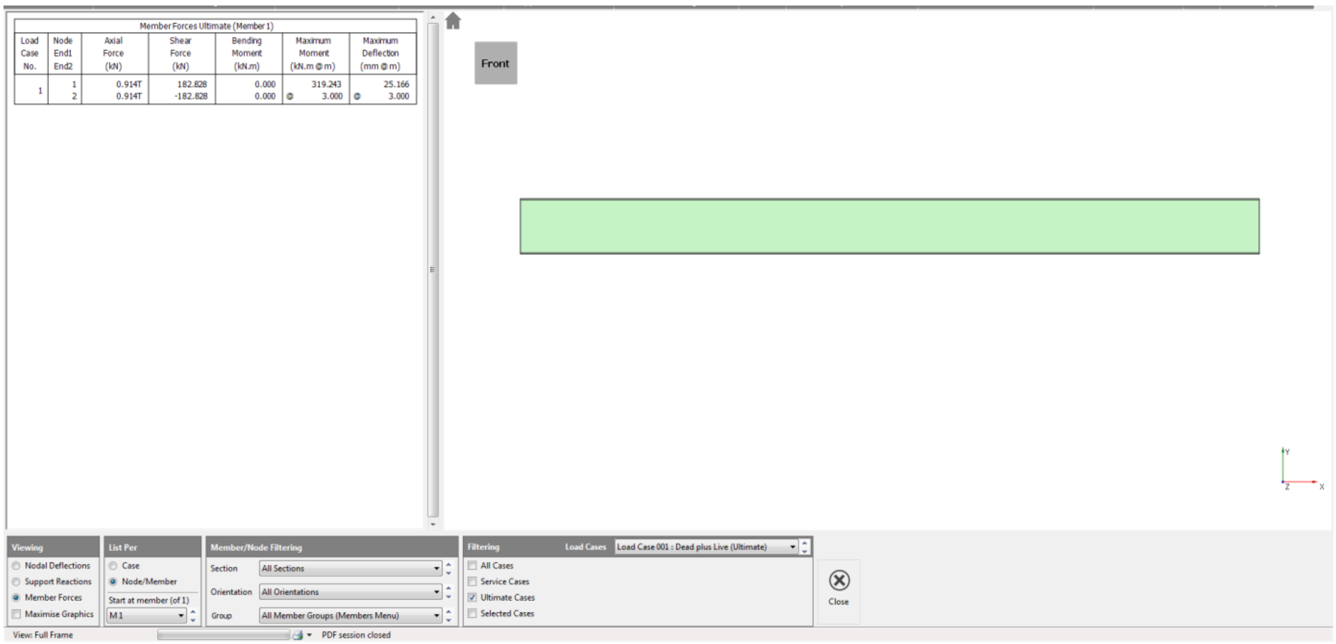
Close the **Job Reference** dialogue by clicking on the **X**.

Select the **Close** button to exit the **Graphical Analysis Output** window.

Viewing Text Results

From the **Results** file menu select **Tabular Analysis Results**. This will take you into the **Nodal Deflections** table.

The options in the lower part of the screen enable you to control the results being displayed and are very easy to use. When there is a large amount of data the vertical scroll bar controls the view.



The standard method of viewing results is **List per Case**. This only display results in one loading case at a time. The other method is to **List per Node/Member**. This is useful to compare results for different loading cases for the same node or member as shown here.

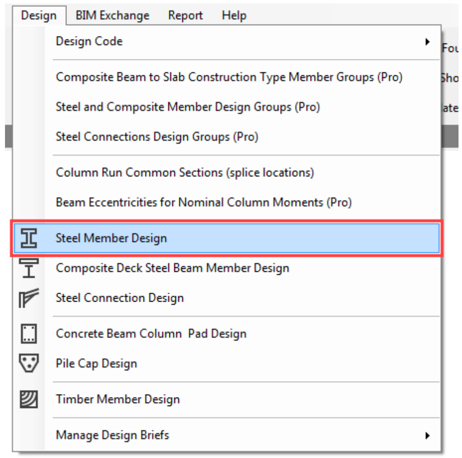
Experiment with the various options to display the results.

To close **Tabular Analysis Results** select the **Close** (
) button.

Steel Design

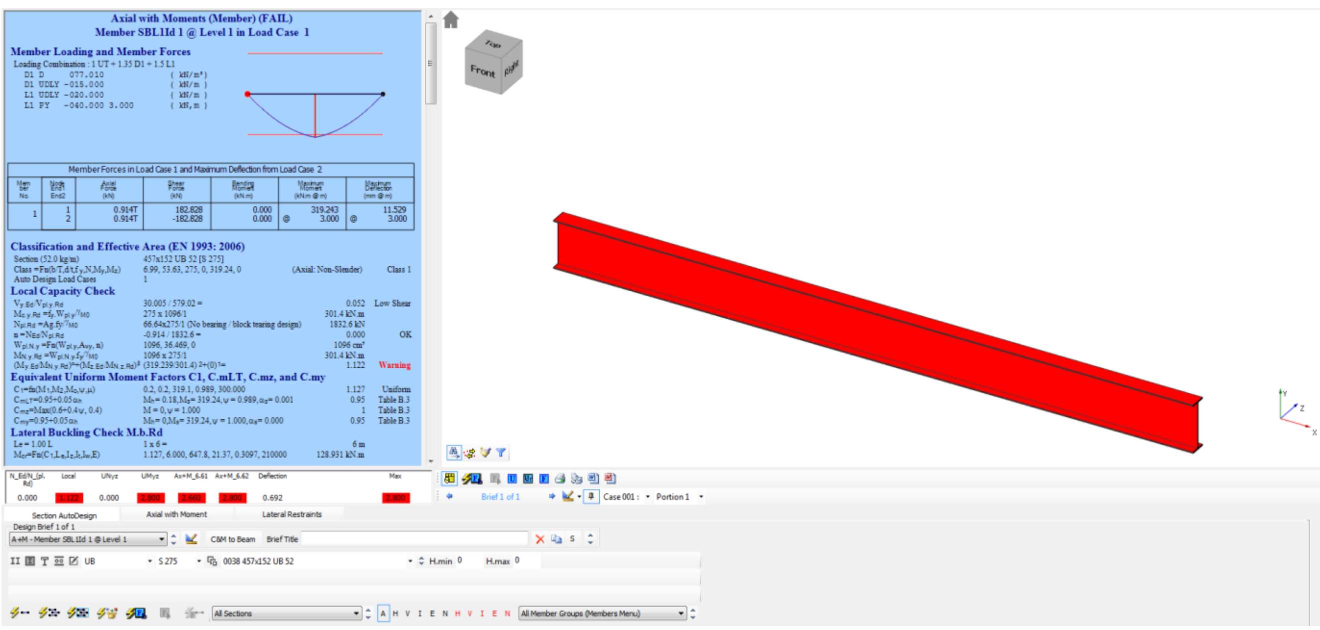
Now we will design the steel beam.

From the **Design** file menu, select the **Steel Member Design** option.



In the case of a single simply supported beam, the program automatically applies an **Axial with Moment** design check to the member.

The results from this check are displayed in the main design output window. The cyan coloured background indicates that there is a design failure.



The tabs at the bottom of the screen contain the information being used for the design check.

The **Section AutoDesign** tab shows the section size and the loading case that is being used for the design. The program automatically detects design cases and serviceability cases. In this case there is only one of each.

Scroll down the design output screen using the scroll bar to the right of it. The results reveal that the beam is failing in both the moment capacity check and the lateral buckling check.

Select the **Axial with Moment** tab. In this tab, you may change fundamental design assumptions such as effective length, deflection limits etc. In most instances the default values are appropriate.

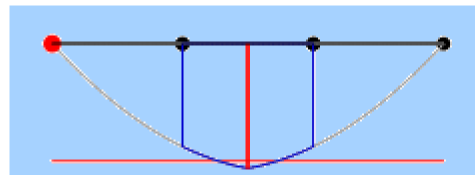
Select the **Lateral Restraints** tab. In this tab, we can define position of lateral restraints along the length of the beam.

In **Portion 1** enter a value of **2.0**.

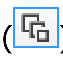
The **Portion 1** entry is actually used as a shortcut for defining an equal spacing of lateral restraints over the entire length of the member. Entering values in further portions overwrites this assumption.

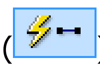
The **Moment Capacity/Bending Moment** diagram at the top of the design output window is now split up into three portions of equal length. The design output relates to the currently selected portion.

Click on the centre portion of the bending moment/capacity diagram to select the middle portion.



Go back to the **Section AutoDesign** tab.

Change the section list to list by weight by selecting the **Sort by Weight** () button when it has a border it is on.

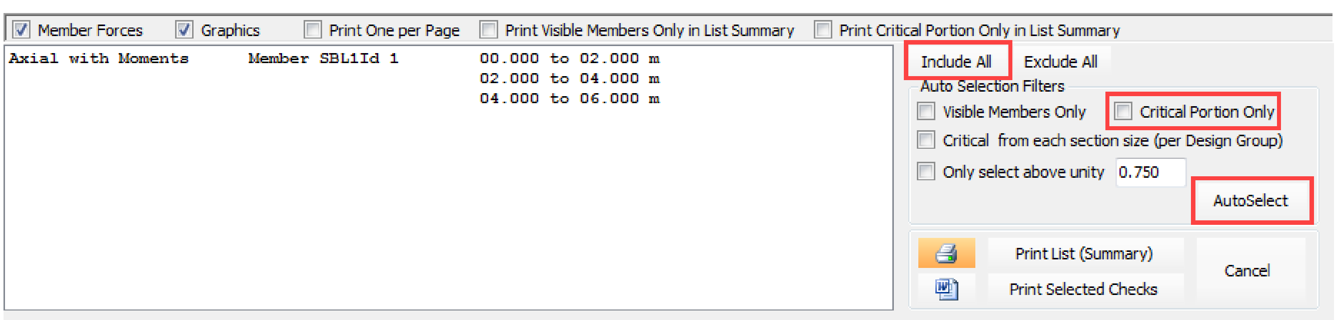
Click on the **Auto Size Current Member** () button.

The program will search for the lightest steel section of the current section type that passes all design checks.

Printing the Design Output

Select **Print Design Output**, from the **Print** menu.

The print manager will appear at the bottom of the screen listing all the design checks applied to the beam. The three portions of the design check are listed separately since each portion has its own set of calculations.



Click on the **Include All** button to highlight all design checks and display their maximum unity ratios.

Check the **Critical Portion Only** checkbox and click on the **AutoSelect** button.

This reduces the currently selected checks to the most critical portion in each member. In this case, only one check is highlighted.

There are two options available:

- Print List (Summary): print the list that is displayed in the print manager window
- Print Selected Checks: print full detail for the design output for the check highlighted in the list

The printed design output appears in exactly the same format as shown on the screen.

Close the **Steel design** by clicking on the main **X**.

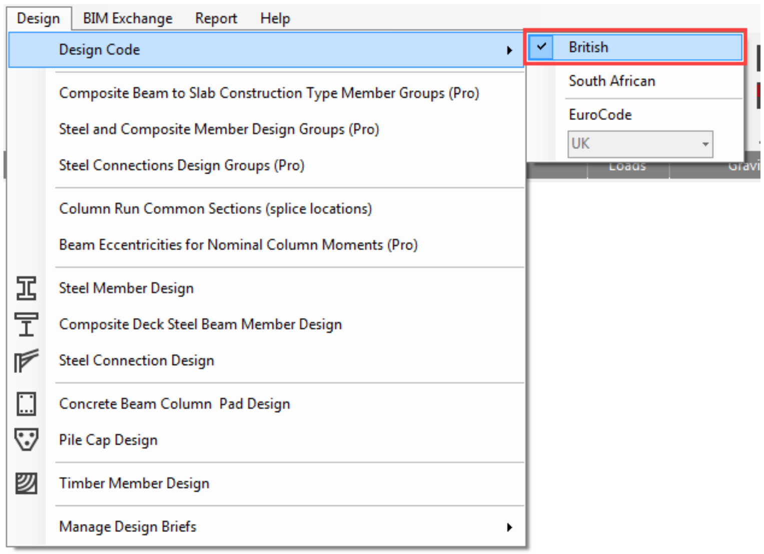
If prompted, save the file.

Redesigning for British Standards

The beam can be redesigned to another code at any stage.

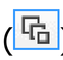
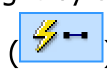
On the top menu, go to **Design > Design Code** and change the selected code to **British**.

This will automatically change the loading combinations to **British Standard**. Now the beam can be re-analysed.



Select **Static Analysis** from the **Analysis** menu and analyse as a **Plane Frame**.

From the **Design** file menu, select the **Steel Member Design** option.

Change the section list to list by weight by selecting the **Sort by Weight** () button and click on the **Auto Size Current Member** () button.

This will automatically redesign the member to **British Standard**. The changes can be seen in the design output. Changes to the Maximum Moment and Deflection, and changes from Plastic to Class 1.

End of Tutorial