

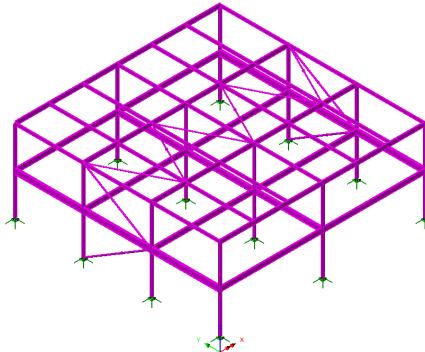
Steel Frame Design

to EN 1993-1-1

For LUSAS version:	16.0
For software product(s):	LUSAS Civil & Structural or LUSAS Bridge.
With product option(s):	Design
Note: This example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example concentrates on the design of steel members of a 3D frame in accordance with EN 1993-1-1:2005+A1:2014. Values for Nationally Determined Parameters (NDPs) are taken from the UK National Annex.



The frame consists of hot rolled structural steel sections, steel grade S 275, hot finished hollow sections, steel grade S 355 H and user-defined grade hollow section.

Columns are spaced at 5m centres along the model's y-axis, and 7.5m centres along its x-axis. The first floor consists of precast slabs spanning 7.5m between the steel beams. A perimeter facade runs around the perimeter of the building and is supported on the first floor perimeter beams. The roof consists of a lightweight deck and appropriate build-up spanning 2.5m between steel beams.

The precast slabs are considered to provide sufficient restraint to the compression flange of the supporting beams at first floor level (and so do not require a check for lateral torsional buckling). All other beams are taken as not having sufficient restraint to the compression flange.

A simplified form of wind load is applied to the intersection of beams and columns in the XZ plane only. The vertical effects of the wind loading are ignored.

The following loads are considered for steel self-weight:

First floor dead load (from precast)	4.2kN/m ²
First floor live load	4.0kN/m ²
Perimeter facade	8.0kN/m
Roof dead load	0.50kN/m ²
Roof live load	0.60kN/m ²
Wind load at 1 st floor level	1.5kN/m ²

The following load combinations are considered:

- Dead + Imposed
- Dead + Imposed (leading) + Wind (accompanying)
- Dead + Wind (leading) + Imposed (accompanying)
- Dead (inf) + Wind

Units used are kN, m, t, s, C throughout.

Objective

- To carry out design checks to a selected design code and confirm suitability for steel member sizes.

Keywords

3D, Frame, Structural Steel, Design code checks, Eurocode 3, EN 1993-1-1, Design Member, Design Attribute, Load Combinations, Enveloping, Error Codes, Member Utilisation Ratio, Member Report, Reporting

Associated Files



steel_frame_design_original_sections.vbs carries out the definition of combined lines (where required) and the definition and assignment of the steel design attributes to all members. To be used where stated in case of user errors in preparing the model for a design check.

Modelling

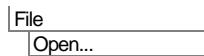
Running LUSAS Modeller

For details of how to run LUSAS Modeller see the heading *Running LUSAS Modeller* in the *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Building and loading the model

For this example a model file is provided:



To load the model, open the read-only file **steel_frame_design_preliminary.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

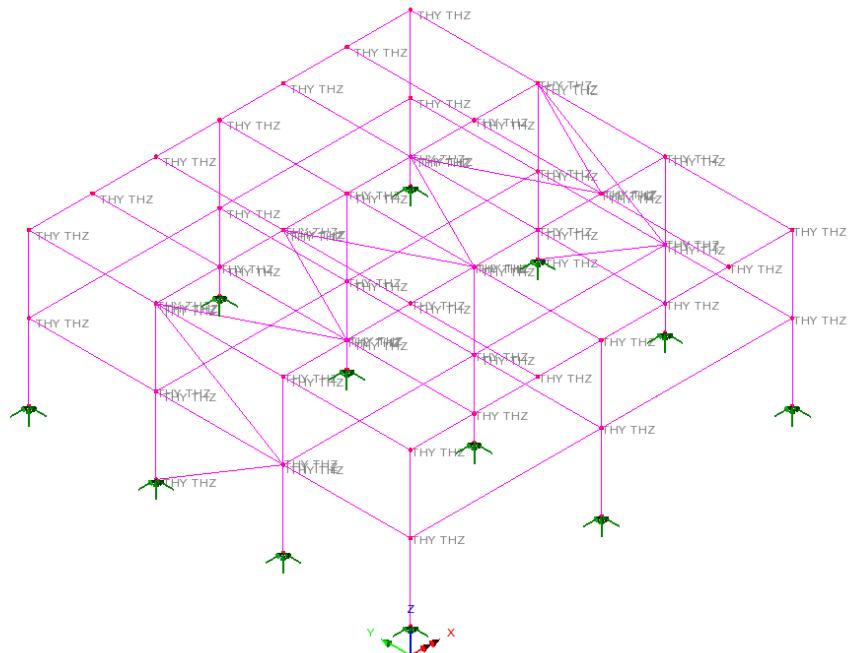
Depending upon your access rights it may be necessary to copy this file to another folder before it can be opened, and remove the read-only protection before it can be saved to the new folder. Once opened:



If necessary, select the isometric button to view the frame in 3D.



Toggling the Fleshing button on and off will show the steelwork arrangement.



Saving the model



Save the model file to the `\<Lusas Installation Folder>\Projects` directory.

Design Code Checking

Design code checking is carried out in LUSAS as a post-processing operation following the solving of an analysis model. In LUSAS a steel member is represented by a line in the model that is assumed to span between and be restrained or supported by its end-points. For situations where a series of lines may represent a single steel member, such as when one beam is supporting cross-beams along its length, the series of lines representing the supporting beam will need to be specified as a combined line by using the combined line facility. Steel beam design attributes will need to be assigned to members to provide design related information. Since the assigned steel beam design attributes have no impact on the structural analysis they can be assigned after the analysis has been run. Changes can be made to a design attribute (for example, changing a grade of steel) without re-running the structural analysis.

Meshing considerations

In the model two line meshes are defined. Each line mesh has different end release conditions and is dependent upon whether the line representing a steel member is considered continuous, has a moment connection at its end or has a simple connection at its end.



Caution. For design code checking purposes a line mesh should have at least 2 elements and preferably (as in this model) a minimum of 4 elements assigned to each line. Using a single element can cause difficulties with detecting the shape of the bending moment diagrams for buckling checks. Using an excessive element assignment to each line will increase the time required to carry out the design checks.



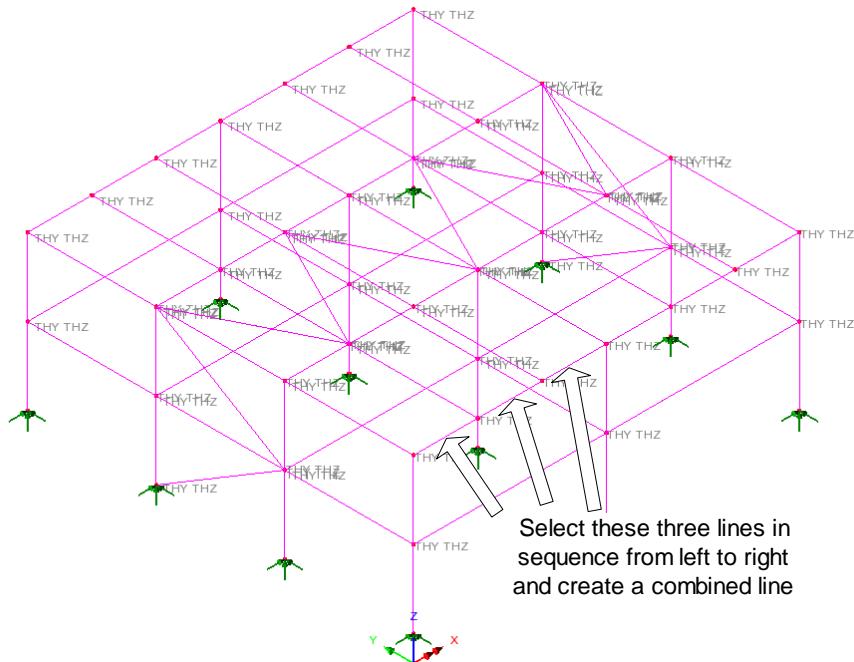
Caution. Checks to a design code cannot (currently) be carried out if the rotation about centroid on the Geometric line attribute is set to anything other than 0 degrees. In the model the external columns were required to be rotated through 90 degrees such that their major axis lay in global x-direction. For design checking purposes this was (and should be) done by setting the beta angle to 90 degrees during the line mesh assignment, rather than by rotating a section about its centroid on the Geometric line attribute dialog. Applying a local coordinate system during the line mesh assignment is an alternative method.

Effects of deformed geometry and whether a first order analysis or second order analysis should be used are not currently considered. Second order effects need to be accounted for manually.

Creating Design Members

Steel members can be represented by either a single line or a combined line (made up of a series of lines). In this example a combined line must be used to define the 7.5m long roof members running parallel to the X-axis between columns. All other members are defined by lines.

- To start, choose the three lines indicated in the order shown on the next image.



Geometry
 Line >
 Combined Line >
 Lines...

 Create the combined line.

- Repeat for the remaining seven sets of three lines representing the roof members running parallel to the X-axis so that eight separate combined lines are created in total.



Caution. The direction of all lines used to define a combined line must be the same. Errors can be introduced in the buckling checks if this is not the case.

Defining Steel Frame Design Attributes

Steel frame design attributes are used to provide additional information about model features in order to enable design checks to supported design codes. Note that this additional information is not required for a general structural analysis to be carried out.

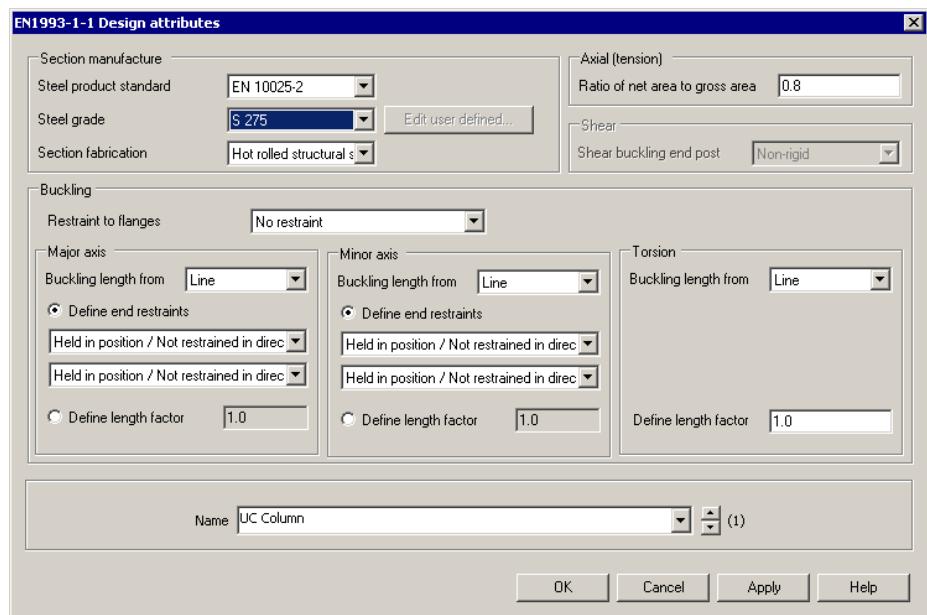
All lines modelling steel members need to be assigned a suitable steel frame design attribute.



Note. Since the design attributes have no impact on a structural analysis they can be assigned after the analysis has been run. Furthermore changes can be made to a design attribute (e.g. changing a grade of steel) without re-running the analysis.

Attributes
Design
Steel Frame
Design...

The steel frame design attribute dialog is displayed:



For the columns:

- With Steel product standard **EN 10025-2** selected, from the **Steel grade** pull-down menu select **S 275**
- Change the **Ratio of net area to gross area** to **0.8**
- Enter the attribute name as **UC column**.
- Click the **Apply** button to add the design attribute to the Attributes treeview.
- Next, in the **Steel product standard** pull-down menu select **EN 10210-1**.
- From the **Steel grade** pull-down menu choose **S 355 H**
- Change the **Ratio of net area to gross area** to **1.0**
- Enter the attribute name as **Hollow column**
- Click the **Apply** button to add the attribute to the treeview.

For the floor beams:

- In the **Steel product standard** pull-down menu select **EN 10025-2**.
- From the **Steel grade** pull-down menu choose **S 275**
- Enter the attribute name as **First floor beam**
- Click the **Apply** button to add the attribute to the  treeview.
- Then, change the **Ratio of net area to gross area** to **0.9**
- Change the **Restraint to flanges** to **Top flange restrained**
- Enter the attribute name as **First floor beam top restrained**
- Click the **Apply** button to add the attribute to the  treeview.

For the roof beams:

- Change the **Restraint to flanges** to **No restraint**
- Enter the attribute name as **Roof beam line**.
- Click the **Apply** button to add the attribute to the  treeview.
- Then, change the **Ratio of net area to gross area** to **1.0**
- For the Major axis change the **Buckling length from** to **Combined line** and for the Minor axis and Torsion, leave as Line.
- Enter the attribute name as **Roof beam combined line**
- Click the **Apply** button to add the attribute to the  treeview.

For the bracing:

- In the **Steel product standard** pull-down menu select **User defined**.
- On the User Defined Grade dialog click in the **Yield stress** cell to cause the units conversion utility arrow  to appear and on the Unit Converter dialog ensure that **N/mm²** is selected, before entering yield stress f_y of **325** and click **OK**.
- In a similar manner, enter an **Ultimate tensile stress f_u** of **450** (**N/mm²**) and click **OK**.
- Enter the **Ultimate strain ϵ_u** as **0.10**
- Enter the **Elongation at failure** as **22 (%)**
- From the **Buckling curve about y-y axis** pull-down menu choose a

- From the **Buckling curve about z-z axis** pull-down menu, choose **a**
- Click **OK**
- On the main Steel Frame Design attribute dialog, in the **Section fabrication** pull-down menu select **Hot finished SHS**
- For the Major axis, change the **Buckling length from** to **Line**.
- Enter the attribute name as **Brace**.
- Click the **OK** button to add attribute to the  treeview and close the dialog.



Note. When the user-defined option is chosen for the Steel product standard, ductility requirements are verified in accordance with EN 1993-1-1 clause 3.2.2 (1). Fracture toughness (3.2.3) and Through-thickness properties (3.2.4) requirements are assumed to have been met.

Assigning the Steel Frame Design Attributes

This is most easily done by assigning the attributes to lines selected by geometric type.

For the columns:

- In the  Treeview right-click on the geometric attribute **External column** and choose **Select Assignments**.
- Drag and drop the design attribute **UC Column** from the  Treeview onto the selected lines.
- Click in a blank part of the Graphics Window to deselect the members.
- In the  Treeview right-click on the geometric attribute **Internal column** and choose **Select Assignments**.
- Drag and drop the design attribute **Hollow Column** from the  Treeview onto the selected lines.
- Click in a blank part of the Graphics Window to deselect the members.

For the floor beams:

- In the  Treeview right-click on the geometric attribute **1st floor beam 3** and choose **Select Assignments**.
- Drag and drop the design attribute **First floor beam** from the  Treeview onto the selected lines.
- Click in a blank part of the Graphics Window to deselect the members.

- In the  Treeview right-click on the geometric attribute **1st floor beam 1** and choose **Select Assignments**.
- In the  Treeview right-click on the geometric attribute **1st floor beam 2** and choose **Select Assignments**. Click **No** from the dialog that appears on screen.
- Drag and drop the design attribute **First floor beam top restrained** from the  Treeview onto the selected lines.
- Click in a blank part of the Graphics Window to deselect the members.

For the roof beams:

- In the  Treeview right-click the geometric attribute **Roof beam 1** and choose **Select Assignments**.
- Drag and drop the design attribute **Roof beam line** from the  Treeview onto the selected lines.
- Click in a blank part of the Graphics Window to deselect the members.
- In the  Treeview right-click the geometric attribute **Roof beam 2** and choose **Select Assignments**.
- Drag and drop the design attribute **Roof beam combined line** from the  Treeview onto the selected lines.
- Click in a blank part of the Graphics Window to deselect the members.

For the bracing:

- In the  Treeview right-click the geometric attribute **Wall Brace** and choose **Select Assignments**.
- In the  Treeview right-click the geometric attribute **Roof Brace** and choose **Select Assignments**. Click **No** from the dialog that appears on screen.
- Drag and drop the design attribute **Brace** from the  Treeview onto the selected lines.

 **Note.** Since the design attributes have no impact on the analysis they can be assigned after the analysis has been run. Furthermore changes can be made to a design attribute (e.g. changing a grade of steel) without re-running the analysis.



Save the model file as **steel_frame_design_original_sections.mdl**

Running the Analysis

With the model loaded:



Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview

Addition files will be created in the directory where the model file resides, including:



- steel_frame_design_original_sections.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- steel_frame_design_original_sections.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

File
Script >
Run Script...

To recreate the model, select the file **steel_frame_design_original_sections.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.



Rerun the analysis to generate the results.

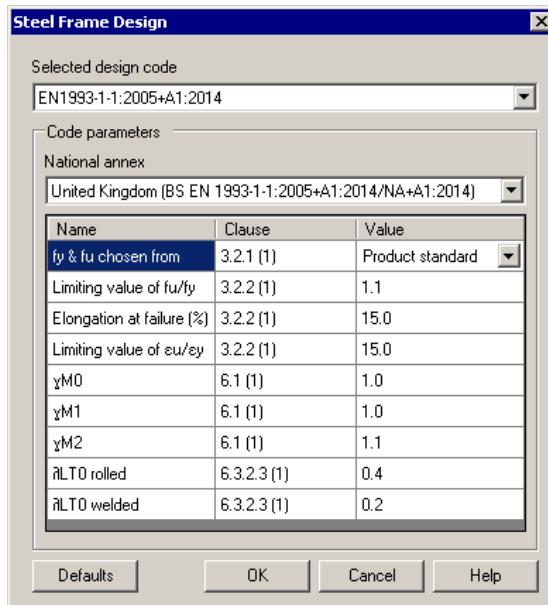
Viewing the Results

Analysis loadcase results are present in the Treeview, and the loadcase results for the last active loadcase are set to be active by default.

- In the Treeview right-click on **Permanent action** and select the **Set Active** option.

Checking against a Design Code

- Choose **EN1993-1-1: 2005+A1:2014** from the **Selected design code** pull-down menu.
- Choose **United Kingdom** from the **National annex** pull-down menu. The NDPs used in the design checks are shown on the dialog.
- Click the **OK** button to finish.

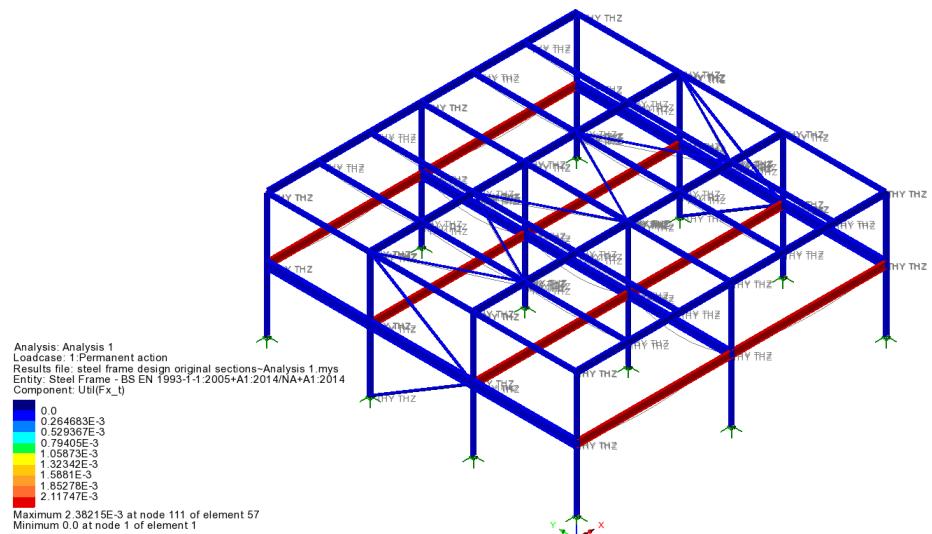


Plotting Results Contours of Utilisation Ratio



Set the Fleshing to 'on' if it isn't already visible.

- With no features selected, click the right-hand mouse button in a blank part of the active window and select the **Contours** option to add the contours layer to the  Treeview.
- Select entity **Steel Frame: BS EN1993-1-1:2005+A1:2004:2014/NA+A1:2014** and component **Util(Fx_t)** from the drop down lists to see utilisation results for all members for the permanent action loadcase.



Defining an Envelope

[Analyses](#)

[Envelope...](#)

An envelope is used to determine the highest utilisation of each member from the four design combinations defined.

On the Properties dialog:

- Select combinations **(5) Dead + imposed**, **(6) Dead + imposed + wind**, **(7) Dead + wind + imposed** and **(8) Dead (inf) + wind** and press the Add to button .
- Change the name to **Design Envelope** and click **OK**.

Two envelope attributes representing the maximum and minimum results will be created in the  Treeview.



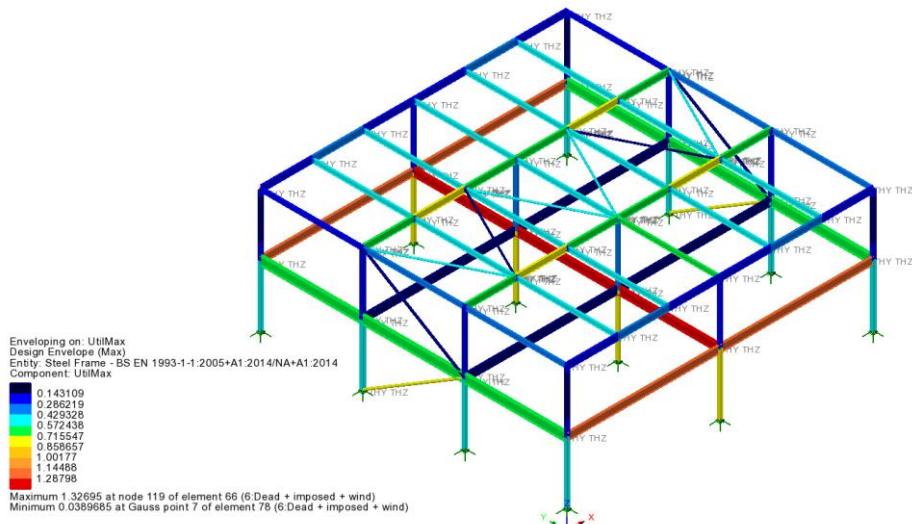
Note. Load combinations (6) and (7) take the imposed load and wind load alternatively as the accompanying load in accordance with EN1990. Different factors are used within each included loadcase.

Selecting and Viewing the Envelope Results

- In the  Treeview right-click on **Design Envelope (Max)** and select the **Set Active** option. On the Set Active dialog leave **Max / min primary component with coincident effects** selected.
- Ensure entity **Steel Frame: BS EN1993-1-1:2005+A1:2004:2014/NA+A1:2014** and component **UtilMax** are selected from the drop down lists

- Select **Change existing drawing layers to above component** (so that contours of UtilMax will be plotted in place of the Util(Fx_t) as previously chosen on the Contours layer) and click the **OK** button.

After a pause whilst design checks are carried out, the utilisation ratios for all members will be shown. It can be seen that the utilisation ratio for a number of members is greater than 1.00. These members have failed the design checks and need to be investigated further.



It can also be seen that a number of members have a very low utilisation ratio. For a refined design these members could also be investigated in order to obtain a more efficient section size, but for this example these members will not be reviewed further.

From the contour plot the load combination causing the maximum utilisation can be seen to be **(6: Dead + imposed + wind)**, as annotated in the contour key.

Member report

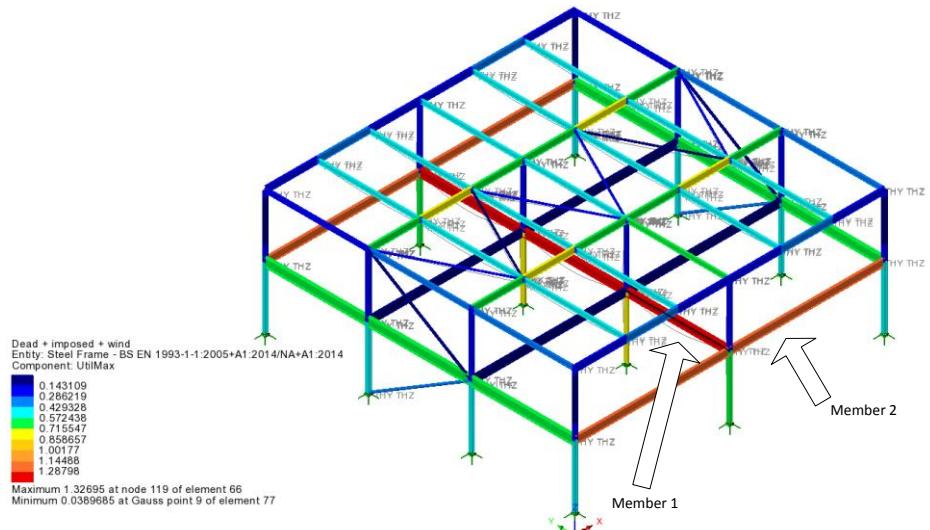
To investigate the design checks carried out for a particular member, a member report can be generated. The values written for each design check in the member report are the most onerous of the results for any (gauss point) position along a member.



Note. A member report cannot be generated for an envelope because co-existent effects would give misleading or incorrect results.

A load case or load combination must be set active before a member report is generated.

- In the  Treeview right-click on **6: Dead + imposed + wind** and select the **Set Active** option. This combination has the same maximum utilisation ratio as that for the **Design Envelope**.



Investigate Member 1

- Select Member 1 as shown on the preceding model image

Design >
Steel Frame Design
Report >
Member...

After a pause whilst design calculations are carried out, a member design report is generated with the Maximum Utilisation (U_{max}) summary displayed. The status column on the left hand side shows the maximum utilisation ratio for each design check made along the line.

Steel Frame Design to EN 1993-1-1

EN1993-1-1:2005+A1:13 design checks

EN1993-1-1:2005+A1:2014 - UtilMax - Maximum utilisation

Member 34: 1st Floor beam 2 (356x171x45kg UB major y)

Element 66, Internal point 1 Utilisation = 0.0

Util(Fx_c) Element = 66, Internal point = 1 Utilisation = 0.0141096

Util(M) Element = 66, Internal point = 11 Utilisation = 1.32695

Util(Fx) Element = 66, Internal point = 1 Utilisation = 0.0

Util(Fz) Element = 66, Internal point = 1 Utilisation = 0.517513

Util(Fy) Element = 66, Internal point = 1 Utilisation = 0.0

Util(Fx,My) Element = 66, Internal point = 1 Utilisation = 0.0

Util(Fy,My) Element = 66, Internal point = 1 Utilisation = 0.0

Util(Fx,My,My) Element = 66, Internal point = 1 Utilisation = 0.0

Util(Fx_c,My,My) Element = 66, Internal point = 1 Utilisation = 0.0

Util(Fx_c,My,My,b) Element = 66, Internal point = 1 Utilisation = 1.32695

UtilMax Element = 66, Internal point = 1 Utilisation = 1.32695

Member 34: 1st Floor beam 2 (356x171x45kg UB major y)

Element 66, Internal point 1 Utilisation = 0.0

Units: kN,m,t,s,C

Design Data

Nominal value of yield strength $f_y = 275.0\text{E}3$ (user)

where f_y is determined from the product standard

Section Properties

Flange thickness $t_f = 9.7\text{E}-3$ (model)

Web thickness $t_w = 7.0\text{E}-3$ (model)

Previously Calculated Parameters

Parameter	Value	Section
Util(Fx,L)	0.0	6.2.3
Util(Fx,c)	0.0141096	6.2.4
Util(My)	0.0	6.2.5
Util(Mz)	0.0	6.2.5
Util(Fz)	0.517513	6.2.6
Util(Fy)	0.0	6.2.6

From the Design Check Summary report it can be seen that the maximum utilisation factor of 1.32695 is obtained from the **Util(Fx_c,My,My,b)** design check.

- Click on the **Util(Fx_c,My,My,b)** entry to review the calculations for this particular design check.

EN1993-1-1:2005+A1:13 design checks

Design Calculation

Utilisation for combined bending and axial compression buckling 6.3.3(4)

$$\text{Util}(F_{x,c}, M_y, M_z, b) = \max(F_{x,c}, M_y, M_z, b_1, F_{x,c}, M_y, M_z, b_2) = \max((1.24594), (1.32695))$$

Combined bending and axial compression buckling check 1 Exp (6.61)

$$F_{x,c}, M_y, M_z, b_1 = \frac{|N_{Ed}|}{\frac{Z_y N_{Rk}}{Y_{M1}}} + k_{xy} \frac{|M_{y,Ed,max} + \Delta M_{y,Ed}|}{\frac{M_y R_k}{\chi_{LT} Y_{M1}}} + k_{yz} \frac{|M_{z,Ed,max} + \Delta M_{z,Ed}|}{\frac{M_z R_k}{Y_{M1}}} = \frac{|(22.2515)|}{(0.953579)(1.57705E-3)} + (0.952769) \frac{|(-275.379) + (0.0)|}{(1.0) \frac{(213.113)}{(1.0)}} + (0.63583) \frac{|(-0.713131E-15) + (0.0)|}{(40.2954)} = 1.32695$$

Moment about y-y axis due to shift of centroidal axis according to 6.2.9.3 $\Delta M_{y,Ed} = 0.0$ Table 6.7

Moment about z-z axis due to shift of centroidal axis according to 6.2.9.3 $\Delta M_{z,Ed} = 0.0$ Table 6.7

Combined bending and axial compression buckling check 2 Exp (6.62)

$$F_{x,c}, M_y, M_z, b_2 = \frac{|N_{Ed}|}{\frac{Z_x N_{Rk}}{Y_{M1}}} + k_{xy} \frac{|M_{y,Ed,max} + \Delta M_{y,Ed}|}{\frac{M_y R_k}{\chi_{LT} Y_{M1}}} + k_{xz} \frac{|M_{z,Ed,max} + \Delta M_{z,Ed}|}{\frac{M_z R_k}{Y_{M1}}} = \frac{|(22.2515)|}{(0.330781)(1.57705E-3)} + (0.993906) \frac{|(-275.379) + (0.0)|}{(1.0) \frac{(213.113)}{(1.0)}} + (1.05972) \frac{|(-0.713131E-15) + (0.0)|}{(40.2954)} = 1.32695$$

- Close the report.



Note. Member design reports may be added as a chapter to a model report by pressing the **Add to report** button that is present in the report listing.

Investigate Member 2

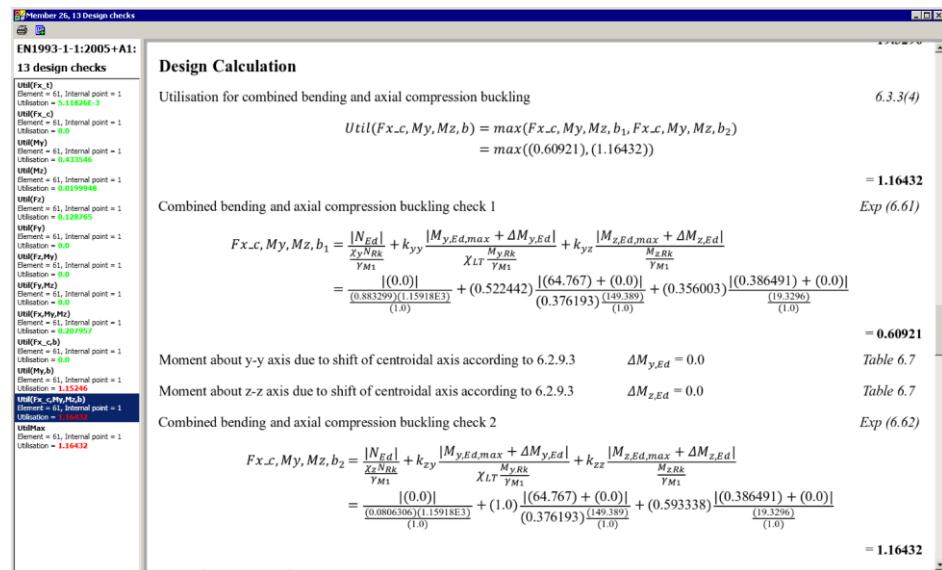
- Select Member 2 as shown on the preceding model image.

Design
Steel Frame Design Report
Member...

After a pause whilst design calculations are carried out, a member design check summary report is generated for this selected member.

From the Design Check Summary report it can be seen that the maximum utilisation factor of 1.16432 is obtained from the **Util(Fx_c,My,My,b)** design check.

- Click on the **Util(Fx_c,My,My,b)** entry to review the calculations for this particular design check.



General notes relating to member reporting

When the design code member report menu item is selected, the initial design check displayed in the member report is that for the active component in the contour plot, and not for the last design check reviewed. If no suitable component is active or no results layer is turned on the member report defaults to the first design check carried out, which in the case of EN1993-1-1 is for tension capacity. If results layers are shown for both contours and diagrams, the contours layer takes precedence.

The **Design> Steel Frame Design Report> Member** menu item generates the

calculations for each design check at the location with the highest utilisation along the line. If the calculations are required to be generated for a specific location along a selected line use the **Design> Steel Frame Design Report> Member (Selected IP)** menu item, and specify the relevant element number and Internal Point.

The entity **UtilMax** shows the highest utilisation ratio in the member. This is associated with a specific element and Internal Point. The utilisation ratios provided in the Design Check Summary for the various design checks are the coexistent values at the specific element and Internal Point.

The **Design> Steel Frame Design Summary** menu item provides a dialog to generate a summary for all or a selection of members. All or just specific design checks can be requested. An option also exists to limit the output to members meeting one of the following criteria:

- Utilisations > 1.0**
- Only errors** Members that cannot be checked due to errors (due to incomplete or incompatible input or if the check is outside the scope of the design checks)
- Only fails** Members that fail. Failure in this case refers to a case where a utilisation ratio is not calculated because a basic criteria is not met e.g. failure of combined shear and flexure check due to shear capacity alone being exceeded.

After making a selection, an entry is created in the Utilities  treeview, which when double-clicked will calculate and display the chosen summary values.

Modifying the model, re-solving and checking results

Changes now need to be made to the model to enable the design to comply with the design code.

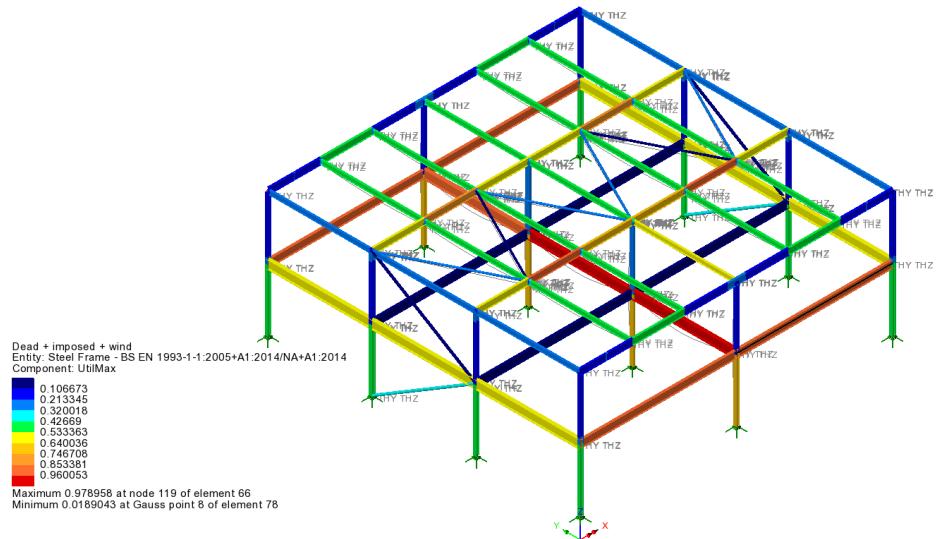
- In the  Treeview double-click on the geometric attribute **1st Floor Beam 2 (356x171x45kg UB major y)**.
- Change the size of the member from 356x171x45kg to **406x178x54kg UB** and click **OK**.

Design results are automatically re-computed to show illustrative effects of making that change to the model. This results in a new maximum value of 1.164 (or similar).

- In the  Treeview double-click on the geometric attribute **1st Floor Beam 3 (356x127x33kg UB major y)**.
- Change the size of the member from 356x127x33kg to **356x127x39kg UB** and click **OK**.

Design results are automatically re-computed to show illustrative effects of making that change to the model. This results in a new maximum value of 0.978 (or similar)

From viewing the results the maximum utilisation is now less than 1.00 meaning that all sections are acceptable.



Tip. To prevent re-calculation of results each time the section properties of a member are updated in this way (as is the default), the active loadcase and the contours layer could be set to show an entity and a component respectively other than that for a design code, or the results file could be closed prior to a re-solve being done once all changes had been made.



Save the model file as **steel_frame_design_revised_sections.mdl**

A re-solve using these revised section properties will confirm this:



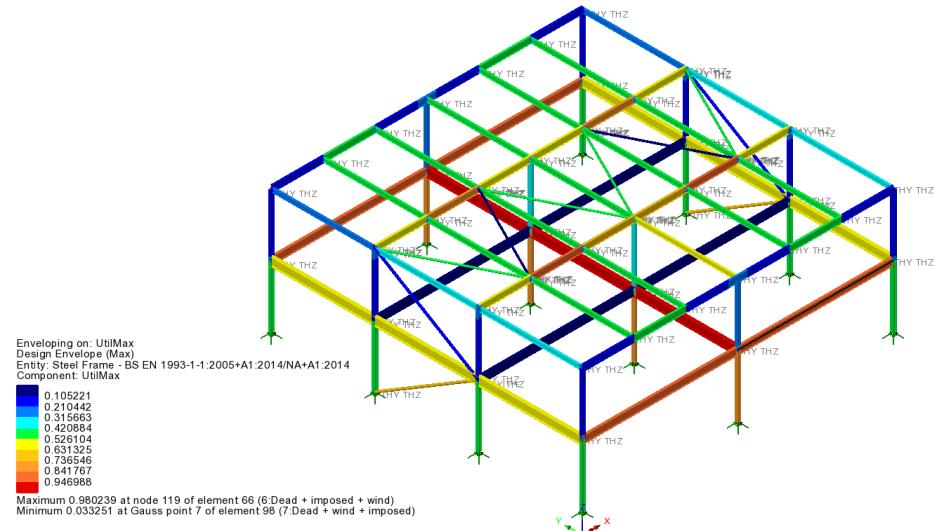
Select the **Solve Now** button to rerun the analysis and generate revised results. This will initially show utilisation contours and maximum and minimum values for the Permanent Action loadcase.

To complete the checking process, utilisation contours for the Design Envelope (Max) will be plotted:

- In the Treeview right-click on **Design Envelope (Max)** and select the **Set Active** option. On the Set Active dialog leave **Max / min primary component with coincident effects** selected.

- Ensure entity **Steel Frame: BS EN1993-1-1:2005+A1:2004:2014/NA+A1:2014** and component **UtilMax** are selected from the drop down lists
- Ensure that **Change existing drawing layers to above component** is selected (so that contours of UtilMax as previously chosen on the Contours layer) and click the **OK** button.

The maximum utilisation is still less than 1.00, meaning that all sections still comply with the design code.



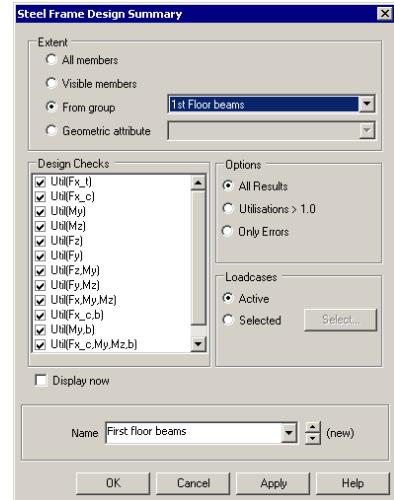
So it can be seen that, for this example, the illustrative results obtained by assigning revised section properties to a previously solved model very closely match the actual results obtained from re-solving the model using those revised properties.

Creating a Steel Frame Design Summary report

Design
Steel Frame Design Summary

From this dialog, selections can be made in order to create a summary of design check results for all or selected members, results and loadcases. To illustrate how a sub-set of results can be output, a previously created group of just the first floor beams is used.

- In the 'From group' drop-down, select **1st Floor beams**
- Ensure 'All' design checks are selected for **All Results** and for the **Active** loadcase.
- Enter a name of **First floor beams**.



On completion a 'First floor beams' entry is created inside the Steel Frame Design Summary folder in the Utilities treeview. If this were to be double-clicked, the chosen steel frame design summary values would be calculated and displayed in a tabular format showing the utilisation of every member for each design check. An example output is shown below. For this example this will not be done. Instead, design summary results will be added to a model report

	Loadcase	Primary	Line	fu	fy	N,u,Rd	N,g,Rd	Nt,Rd	N,Ed	Util(Fx,_I)	Comments
1	Loadcase	Primary	Line	fu	fy	N,u,Rd	N,g,Rd	Nt,Rd	N,Ed	Util(Fx,_I)	Comments
2	9 Design Envelope	Fx	25	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78803	4.22723E-3	
3	9 Design Envelope	Fx	26	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78803	4.22723E-3	
4	9 Design Envelope	Fx	27	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.19865	5.86795E-3	
5	9 Design Envelope	Fx	28	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.19865	5.86795E-3	
6	9 Design Envelope	Fx	29	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.25788	6.03106E-3	
7	9 Design Envelope	Fx	30	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.25788	6.03106E-3	
8	9 Design Envelope	Fx	31	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78802	4.22722E-3	
9	9 Design Envelope	Fx	32	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78802	4.22722E-3	
10	9 Design Envelope	Fx	33	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
11	9 Design Envelope	Fx	34	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
12	9 Design Envelope	Fx	35	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
13	9 Design Envelope	Fx	36	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
14	9 Design Envelope	Fx	37	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
15	9 Design Envelope	Fx	38	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
16	9 Design Envelope	Fx	39	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
17	9 Design Envelope	Fx	40	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
18	9 Design Envelope	Fx	41	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
19	9 Design Envelope	Fy	25	410.0E3	275.0E3	1.67023E3	1.36923E3	5.12265	3.74127E-3		
20	9 Design Envelope	Fy	26	410.0E3	275.0E3	1.67023E3	1.36923E3	5.78802	4.22722E-3		

Adding steel design summary reports to main reports



Note. Steel member design summary report data may be added as a chapter to a model report in three ways:

1. By pressing the **Add to report** button that is present on the report listing.

2. By selecting the **Add to report** option from the context menu of the 'First floor beams' entry in the Utilities  treeview.
3. By creating a new report and including the name of the entry in the Utilities  treeview that creates the summary results.

The last method is used in this example.

On the report dialog enter a report title of **Steel frame design**.

- Leave the modelling units and the number of significant figures set to default values.
- Enter a report name of **Steel frame design** and click **OK**.

A report entry will be created in the Report Treeview.

- In the  Treeview right-click on **Steel frame design** and select **Add Chapter**.
- Choose the **Results** tab.
- Check (tick) to **Include** the utility named **First floor beams** and click **OK**.
- In the  Treeview right-click on **Steel frame design** and select **View Report**.

A PDF of the report is created

This can be saved and printed as required.

Save the model

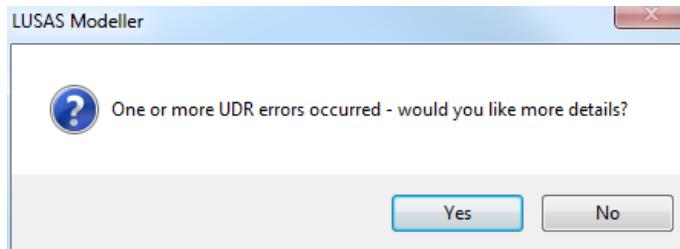


Save the model file.

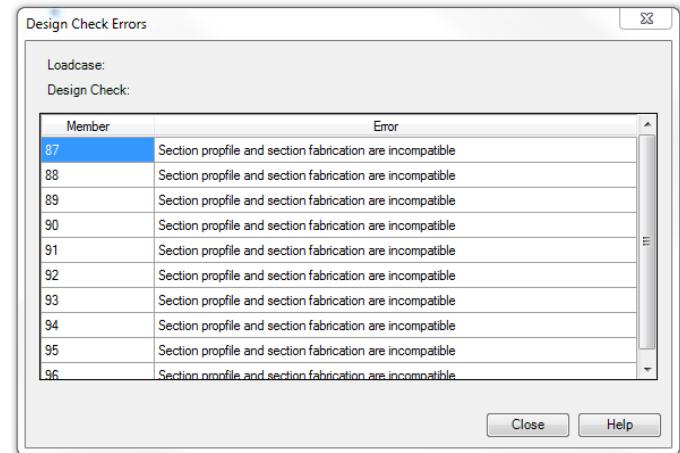
This completes the example.

Discussion - error checking

If there are errors in the model that prevent design checks from taking place the following dialog will appear. To find out more details click **Yes** on the dialog.



The following dialog will appear.



This dialog identifies all members affected by errors. When a member or error in the dialog is active, that member can be selected on the model by right clicking on the dialog and clicking **Select member**. In this discussion example the error message indicates an issue with the section profile and the section fabrication, requiring a different method of fabrication to be selected.

