Steel Frame Design to EN 1993-1-1

For LUSAS version:	19.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	Steel Frame Design

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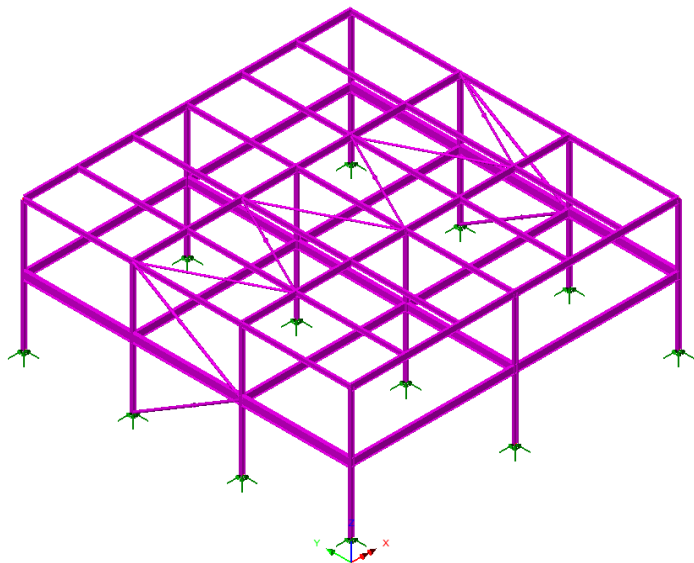
The frame consists of hot rolled structural steel sections, steel grade S 275, hot finished hollow sections, steel grade S 355 H and user-defined grade hollow section.

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 in addition to the self-weight of the steel beams:

Location	Loading
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 1st 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, Steel, Frame, Structural, 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_preliminary.mdl** contains the basic model geometry. The example will use this model as a starting point.

❑ **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 an initial model file is provided:

File
Open...

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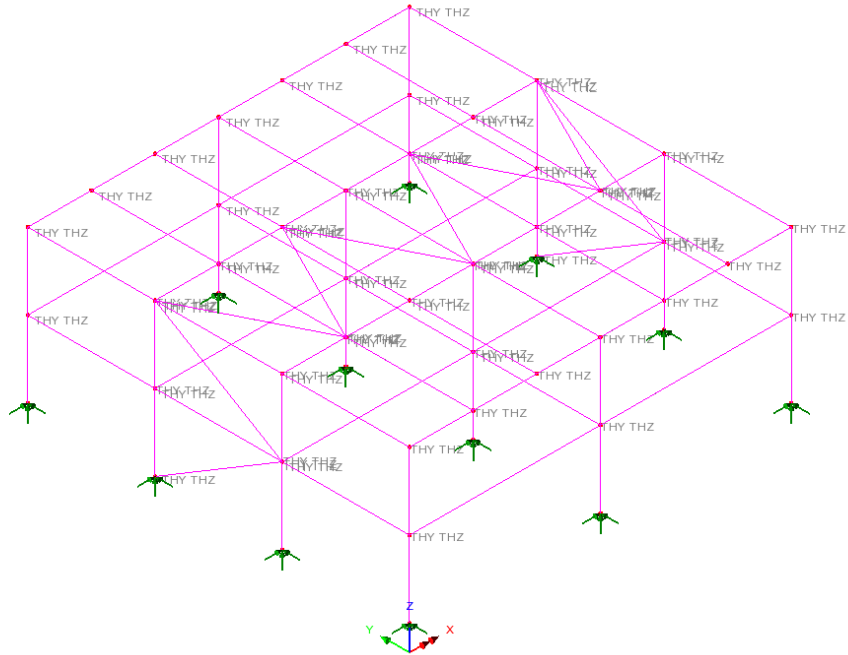
If necessary, select the isometric button to view the frame in 3D.

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 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 using thick beam element with a linear interpolation order so that the example can be run in any LUSAS software product version. Otherwise the use of quadratic elements is recommended. 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, or if a section has been mirrored prior to being assigned to a model. In the model supplied, 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.

- Select the three lines indicated in the order shown on the next image.

Select these three lines in
sequence from left to right
and create a combined line
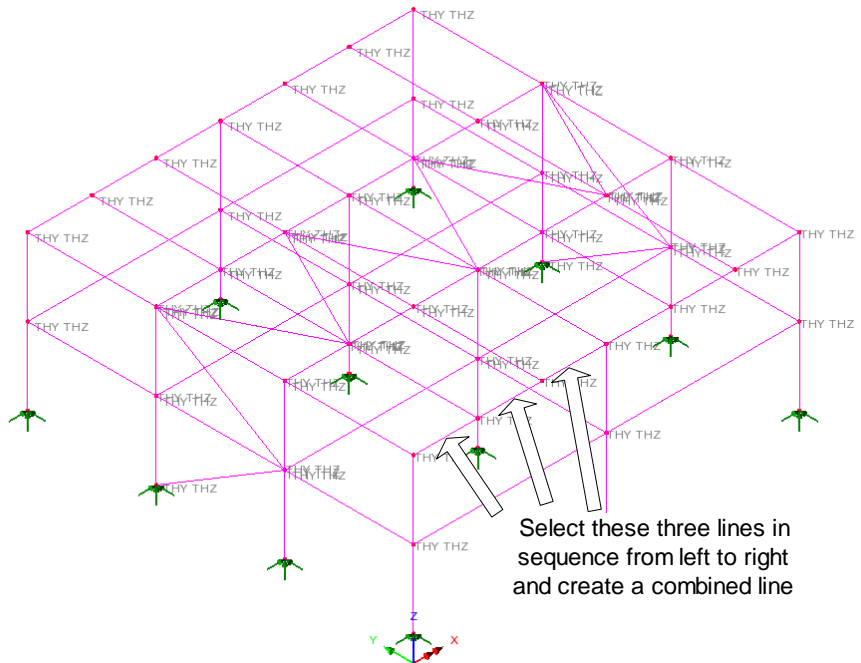
- Select the three lines indicated in the order shown on the previous image.

Geometry	
Line	>
Combined Line	>
Lines...	

This creates a combined line from the selection. The line colour changes to indicate a combined line.

- **Repeat this procedure** 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 the 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. However, first a design code needs to be selected.

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

Selecting a Design Code

Design
Steel Frame
Design...

- 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.

Steel Frame Design		
Selected design code		
EN1993-1-1:2005+A1:2014		
Code parameters		
National annex		
United Kingdom (BS EN 1993-1-1:2005+A1:2014/NA+A1:2014)		

Name	Clause	Value
Class 1 & 2 bending...	5.4.2	Plastic section ...
fy & fu chosen from	3.2.1 (1)	Product standard
Limiting value of fu/fy	3.2.2 (1)	1.1
Elongation at failure...	3.2.2 (1)	15.0
Limiting value of εu...	3.2.2 (1)	15.0
γM0	6.1 (1)	1.0
γM1	6.1 (1)	1.0
γM2	6.1 (1)	1.1
λLT0 rolled & hollow	6.3.2.3 (1)	0.4

Defaults   OK   Cancel   Help

Next create the design attributes.

Attributes
Design
Steel Frame
Design...

The steel frame design attribute dialog is displayed:

For the columns:

- Ensure that 'Section fabrication' is set to **Hot rolled section**

- Ensure that 'Steel product standard' **EN 10025-2** is 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 fabrication' pull-down menu select **Hot finished SHS**.

- In the 'Steel product standard' pull-down menu ensure **EN 10210-1** is selected.

- 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 fabrication' pull-down menu select **Hot rolled section**.

- Ensure that 'Steel product standard' **EN 10025-2** is selected.
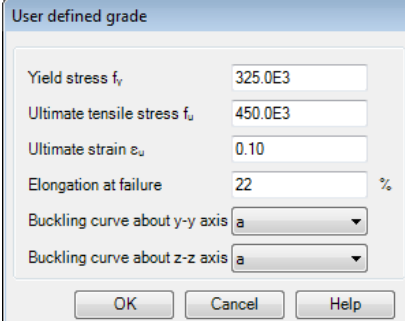
- From the 'Steel grade'pull-down menu choose **S 275**

- 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**

- Seelct the **Bending** tab, and change the 'Flange restraint' 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:

- Staying on the **Bending** tab, change the **Flange restraint** to **No restraint**

- Enter the attribute name as **Roof beam line**.

- Click the **Apply** button to add the attribute to the 🔧 treeview.

- Switch to the **General** tab and change the **Ratio of net area to gross area** to **1.0**

- Switch to the **Compression** tab and for the Major axis change the 'Unbraced length from**'** to **Combined line** but for the Minor axis and Torsion settings, 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:

- Switch to the **General** tab and in the **Section fabrication** pull-down menu select **Hot finished SHS**

- 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 [icon] to appear and on the Unit Converter dialog ensure that **N/mm^2** 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/mm^2) 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**

- Switch to the **Compression** tab, and for the Major axis, change the 'Unbraced length from' setting 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.

- In the 🔴 Treeview right-click on the geometric attribute **Internal column** and choose **Select Assignments**. Ensure **Clear selection** is chosen on the Options dialog, and click **OK**.

- Drag and drop the design attribute **Hollow Column** from the 🔴 Treeview onto the selected lines.

For the floor beams:

- In the 🔴 Treeview right-click on the geometric attribute **1st floor beam 3** and choose **Select Assignments**. Ensure **Clear selection** is chosen on the Options dialog, and click **OK**.

- Drag and drop the design attribute **First floor beam** from the 🎲 Treeview onto the selected lines.

- In the 🎲 Treeview right-click on the geometric attribute **1st floor beam 1** and choose **Select Assignments**. Ensure **Clear selection** is chosen on the Options dialog, and click **OK**.

- In the 🎲 Treeview right-click on the geometric attribute **1st floor beam 2** and choose **Select Assignments**. Ensure **Add to selection** is chosen on the Options dialog, and click **OK**.

- Drag and drop the design attribute **First floor beam top restrained** from the 🎲 Treeview onto the selected lines.

For the roof beams:

- In the 🎲 Treeview right-click the geometric attribute **Roof beam 1** and choose **Select Assignments**. Ensure **Clear selection** is chosen on the Options dialog, and click **OK**.

- Drag and drop the design attribute **Roof beam line** from the 🎲 Treeview onto the selected lines.

- In the 🎲 Treeview right-click the geometric attribute **Roof beam 2** and choose **Select Assignments**. Ensure **Clear selection** is chosen on the Options dialog, and click **OK**.

- Drag and drop the design attribute **Roof beam combined line** from the 🎲 Treeview onto the selected lines.

For the bracing:

- In the 🎲 Treeview right-click the geometric attribute **Wall Brace** and choose **Select Assignments**. Ensure **Clear selection** is chosen on the Options dialog, and click **OK**.

- In the 🎲 Treeview right-click the geometric attribute **Roof Brace** and choose **Select Assignments**. Ensure **Add to selection** is chosen on the Options dialog, and click **OK**.

- Drag and drop the design attribute **Brace** from the 🎲 Treeview onto the selected lines.

File
Save As

💾 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.

Rebuilding a Model

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.
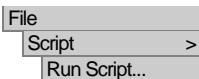
Viewing the Results

Analysis loadcase results are present in the 🕒 Treeview.

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

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 deselect the **Deformed mesh** option.

- 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 and click **OK** to see utilisation results for all members for the permanent action loadcase.

Analysis: Analysis 1
Loadcase: 1:Permanent action
Results file: steel_frame_design_original_sections~Analysis 1.mys
Entity: Steel Frame - BS EN 1993-1-1:2005+A1:2014:2014/NA+A1:2014
Component: Util(Fx_t)

0.0
0.263783E-3
0.527565E-3
0.791348E-3
1.05513E-3
1.31891E-3
1.5827E-3
1.84648E-3
2.11026E-3

Maximum 2.37404E-3 at node 73 of element 77
Minimum 0.0 at node 1 of element 1
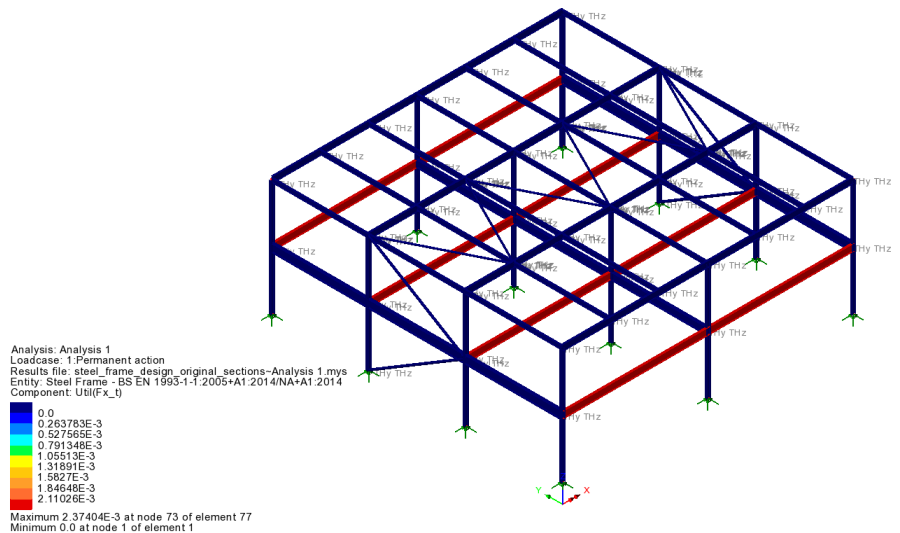
Defining an Envelope

| Analyses |
| Envelope… |

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

In the Available panel of the Envelopes 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 ensure that **Max / min primary component with coincident effects** is 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.

Design calculations will take place. The progress bar at the bottom of the interface shows an approximation of the time remaining to complete the task.

Note. Results for Modeller-calculated results components (such as those calculated for combinations and envelopes, and also for other derived components such as design code-based utilisations, Wood-Amer, user-defined results etc.) can be stored in order to speed-up the initial and subsequent display of those results. Click **OK** (**Yes**) on the dialog that appears to do this. Click **OK** on the subsequent dialog also.

Once calculated, 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 will be investigated further.

Note that the node referenced by the contour key may be different from that shown.

Enveloping on: UtilMax
Design Envelope (Max)
Entity: Steel Frame - BS EN 1993-1-1:2005+A1:2014/NA+A1:2014
Component: UtilMax

0.143109
0.286218
0.429328
0.572437
0.715546
0.858655
1.00176
1.14487
1.28798

Maximum 1.32695 at node 59 of element 66 (6:Dead + imposed + wind)
Minimum 0.0389706 at Gauss point 7 of element 78 (6:Dead + imposed + wind)
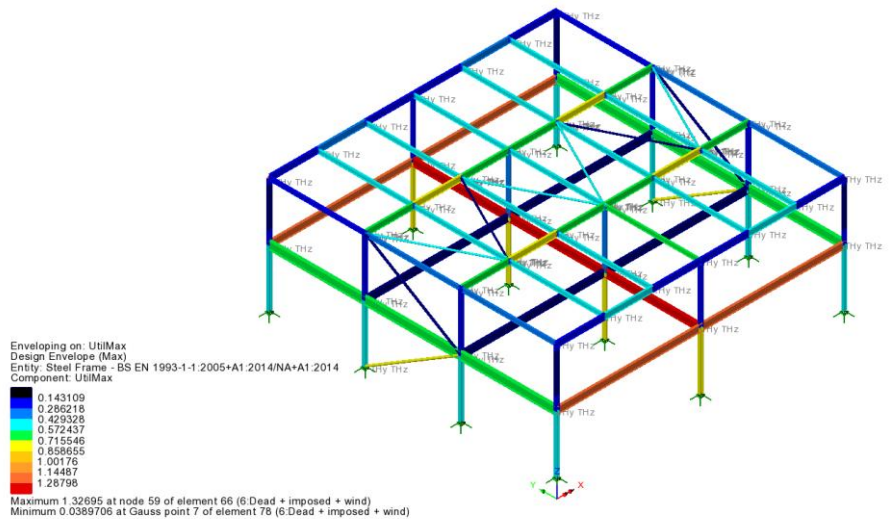
It can also be seen that a number of members have a very low utilisation ratio. For a refined design these members could also be investigated in order to obtain a more efficient section size, but for this worked example these members will not be reviewed further.

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**.

15
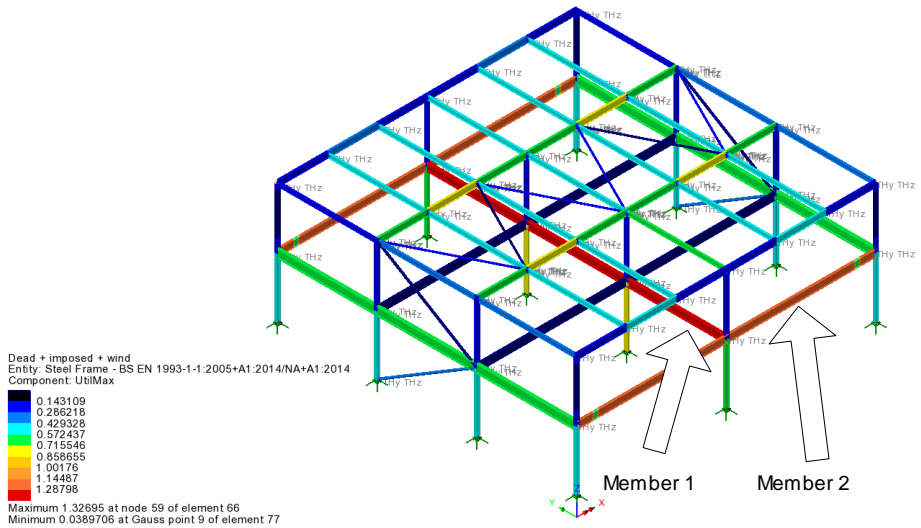
Member 1   Member 2

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.

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,Mz,b)** design check, which can be found at the bottom of the listing.

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

After a pause whilst design calculations are formatted, the results for this design check will be displayed.

<div style="display:flex; align-items:center; gap:8px">

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.

</div>

- Close the member report.

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 1.19975 is obtained from the **Util(Fx_c,My,Mz,b)** design check.

- Click on the **Util(Fx_c,My,Mz,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 Maximum Utilisation summary. 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 Results** 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:

❑ **All Results**

❑ **Utilisations > 1.0**

❑ **Only Messages** Members that cannot be checked due to the messages stated – either due to incomplete or incompatible input, or if the check is outside the scope of the design checks.

After making a selection, an entry is created in the Utilities 🔧 treeview. The summary values can be displayed by double-clicking the entry and ensuring that **Display now** is selected.

Modifying the model, re-solving and checking results

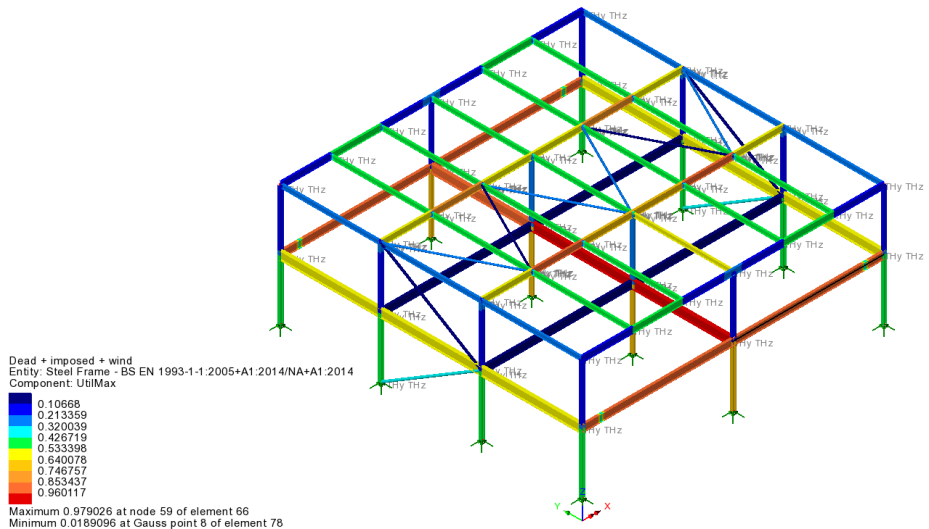Changes now need to be made to the model to enable the design to comply with the design code.

- 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 1.199 (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 0.979 (or similar)

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

Dead + imposed + wind
Entity: Steel Frame - BS EN 1993-1-1:2005+A1:2014/NA+A1:2014
Component: UtilMax

0.10668
0.213359
0.320039
0.426719
0.533398
0.640078
0.746757
0.853437
0.960117

Maximum 0.979026 at node 59 of element 66
Minimum 0.0189096 at Gauss point 8 of element 78

Tip. If it is required to stop re-calculation of results each time the section properties of a member are updated in this way (as is the default), turn-off the display of any of the active results layers (contours, values, diagrams etc.) or close the results file prior to requesting a re-solve being done once all changes have been made.

File
Save As

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

A re-solve using these revised section properties will confirm this:
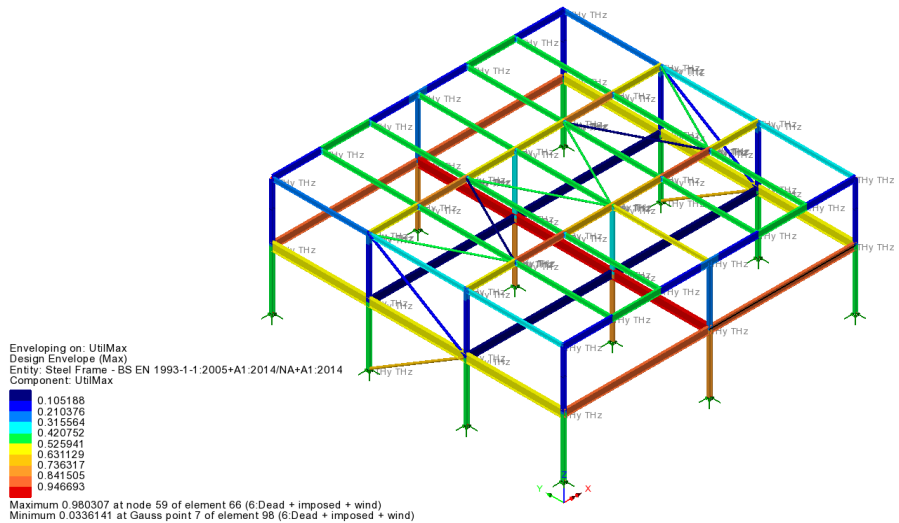
Select the **Solve Now** button to rerun the analysis and generate revised results. This will initially show utilisation contours and maximum and minimum values for the Permanent Action loadcase.

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, ensure that **Max / min primary component with coincident effects** is 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.


Enveloping on: UtilMax
Design Envelope (Max)
Entity: Steel Frame - BS EN 1993-1-1:2005+A1:2014/NA+A1:2014
Component: UtilMax

0.105188
0.210376
0.315564
0.420752
0.525941
0.631129
0.736317
0.841505
0.946693

Maximum 0.980307 at node 59 of element 66 (6:Dead + imposed + wind)
Minimum 0.0336141 at Gauss point 7 of element 98 (6:Dead + imposed + wind)

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
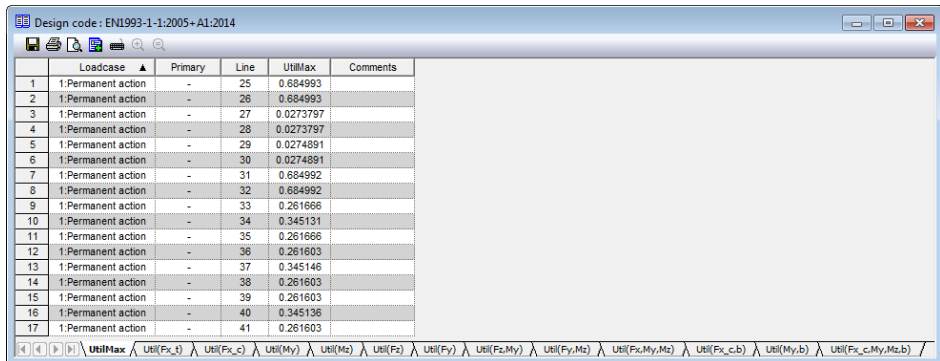
Design
Steel Frame Design
Results

From this dialog, selections can be made in order to create a summary of steel frame 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**

- In the design checks panel, ensure '**All'** available design checks are selected, for **All Results**, and for the **Active** loadcase.

- Enter a name of **First floor beams**.

- Click **OK**.

21
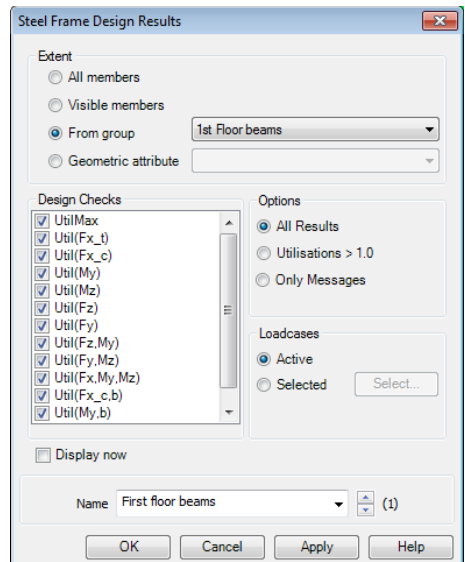
On completion a 'First floor beams' entry is created inside the Steel Frame Design Results folder in the Utilities 🔧 treeview.

If the **Display now** context menu item were selected for the 'First floor beams' entry, 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.

Once done, steel frame design results could be added to a model report by using the 'Add to report button' 📊 on the results dialog, but for this example this will not be done. Instead, steel frame design summary results will be added to the model report using the reporting facility directly.

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 (as described in the previous section).

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 to be used in this example.

Utilities
   Reporting
      New Report...

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**.

After a period of time a PDF of the report is created. This can be saved and printed as required.
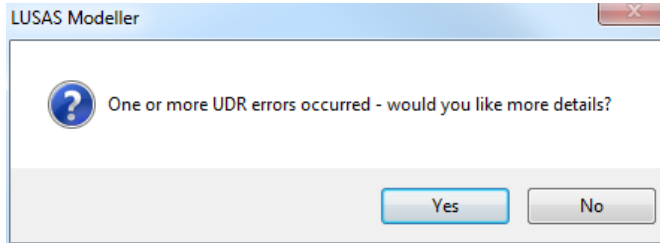
Save the model
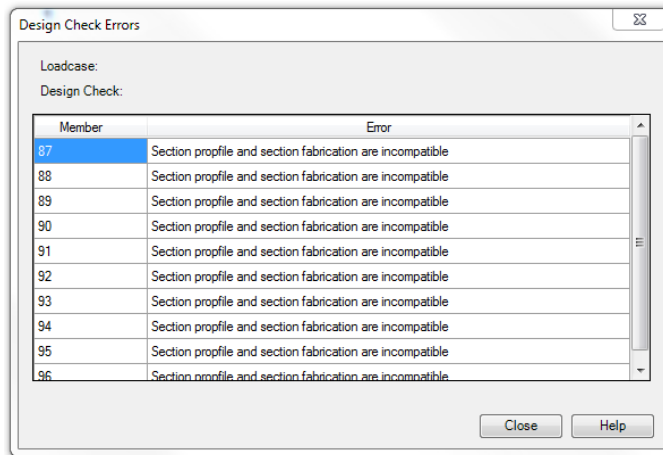
File
  Save

💾 Save the model file.

This completes the example.

Discussion on error checking

If there are errors in a 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, identifying 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 example dialog, the error message indicates an issue with the section profile and the section fabrication, and requires a different method of fabrication to be selected.