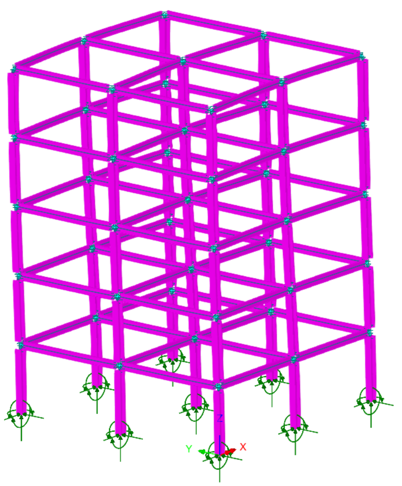
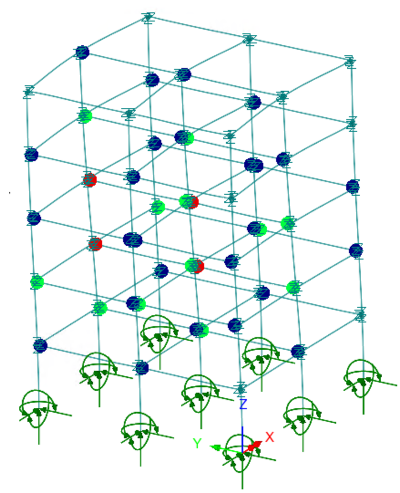
Pushover Analysis of a Steel Frame

For LUSAS version:	24.0
For software product(s):	All (except LT versions)
With product option(s):	Nonlinear.

Description

This example shows how to carry out a pushover analysis of a simple steel framed building according to Eurocode 8. Automatic steel pushover hinges are used to simplify input. True force-deformation curves are investigated to verify their definition. The loading applied is proportional to the critical eigenvalue mode. Nonlinear settings are adjusted to get the best results and avoid convergence issues. The pushover curve is post-processed to determine the performance point. Finally, the hinge distribution across the model at total load factor and at target displacement is investigated and the critical joint response inspected.





Units used are N, m, kg, s, C throughout.

Objectives

The primary objective of this study is to:

- **Define analyses** – showing how eigenvalue and pushover analyses can be created.
- **Apply pushover loads** – showing how a load proportional to mass and fundamental eigenmode can be added to the model.
- **Hinge definition** – showing how steel plastic hinges can be defined and assigned to beams and columns. Their true-force deformation curves are inspected as well.
- **Adjust nonlinear settings** – Pushover analysis includes significant deformations and softening. Therefore, guidance on Nonlinear & Transient control parameters is presented.
- **Extract pushover curve**– A results processing tool to extract the pushover curve is presented.
- **Determine target displacement** – The performance point / target displacement is determined using a Eurocode 8 procedure.
- **Investigate hinges** – The spread of plastic hinges at target displacement and throughout the structure is investigated. The response of a critical hinge is investigated.

Keywords

Force-deformation curves, Inspect hinges, Performance point, Plastic hinges, Pushover analysis, Target displacement, Prior Results-Based Variation.

Associated Files

Associated files can be downloaded from the user area of the LUSAS website.

- **pushover_steel.lvb** creates an initial model for further development.

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.

Creating a New Model

File
New…

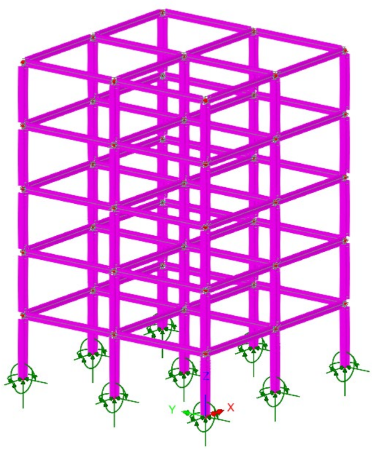
- Enter a file name of **pushover_steel**.
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **3D** is set.
- Click the **OK** button.

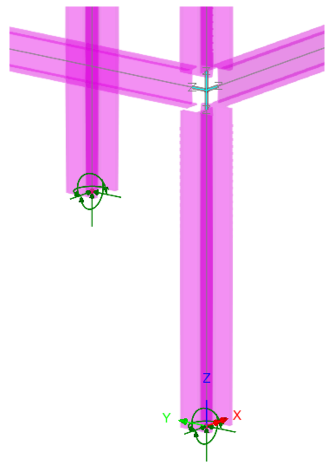
Note. There is no need to enter any other new model details when the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script…

To create the model, open the read-only file **pushover_steel.lvb** that was downloaded and placed in a folder of your choosing.

A simple 5-storey steel moment frame is created, with the major axes of the columns lying in the global X direction.





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

Toggling the Fleshing button on and off as well as using the Fleshing all transparent button will show the steelwork arrangement and orientation of the members.

Base analysis

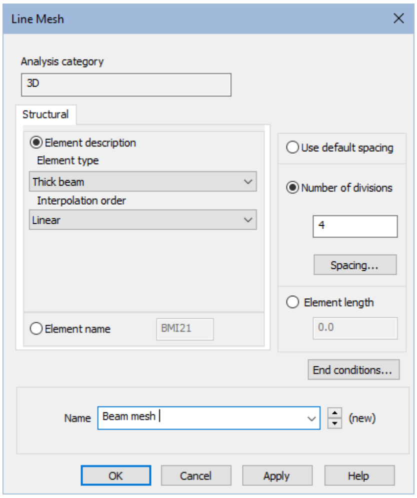
Note. In the Analysis Treeview, loadcase 1 is already present. When modelling, it is good keep practice to retain Analysis 1 as the base analysis (a basic linear analysis) that can be used to check the model is working correctly, meaning that reactions/displacements can be checked prior to more detailed analysis being undertaken. Separate analyses will be added to investigate Modal and Pushover effects

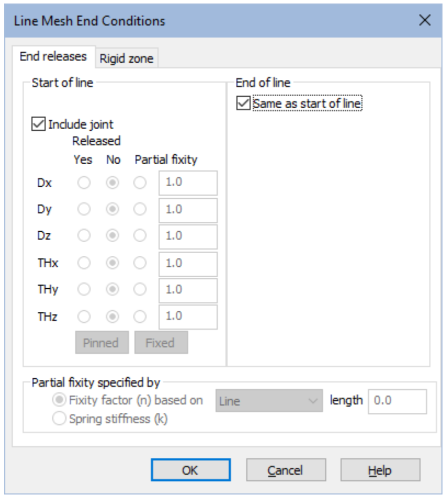
Line beam mesh end conditions

To model pushover the line mesh for the lines representing beam and columns will need to include joints. Plastic hinges will be assigned to these lines, so the beam and column mesh end conditions need to be modified.

Beams

- In the Attributes treeview double click on **Beam mesh** and on the resulting dialog click **End conditions…**

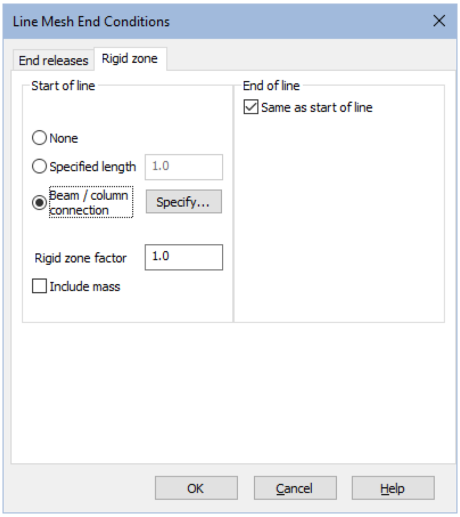


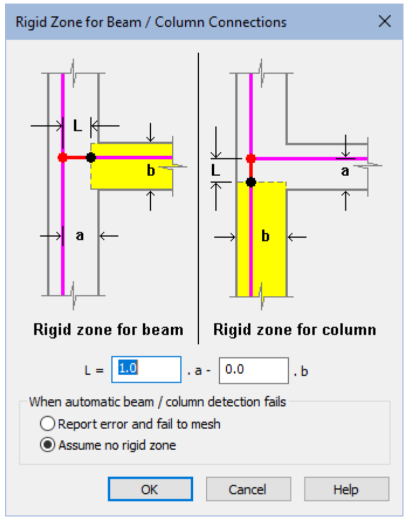


- On the Line Mesh End Conditions dialog, and on the **End releases** tab, tick **Include joint** for start of line, and ensure that **Same as start of line** is ticked for end of line.

Rigid links will be defined at the element ends. These simulate the reduced element length at the beam-column or beam-beam intersections.

- With the Line Mesh End Conditions dialog still active, select the **Rigid zone** tab.





- Select **Beam / column connection** for the start of line and ensure that **Same as start of line** is selected for the end of line.
- Pressing the **Specify** button shows that the length of the rigid zone will be automatically evaluated to account for the column width the beam is attached to.
- Click **OK** as necessary to update the attribute.

Columns

- In the Attributes treeview double click on **Column mesh** and repeat the same steps to include a joint and a rigid zone.
- On the rigid zone tab ensure that the option to **Include mass** is also chosen and click **OK**.

Note. The use of automatic rigid links at beam/column intersections may produce errors around diagonal bracing members, as used in steel frames. Members at 45 degrees, can prevent the model from automatically determining the rigid link. In this case, a 'Specified length' of the rigid zone must be used instead.

Loading

Pushover loads are applied on the structure as body force acceleration. As such, it is important to model gravitational loads using mass. **Non-structural mass** elements can be used (but these would require equivalence attributes to be defined and assigned). In this example a simplified method is used.

The load applied to the structure has already been replicated by artificially increasing the density of the steel beams. This can be inspected by opening the **Steel + loading** entry in the Attributes treeview, for which the density is set to 500e3). This captures the slab weight and accidental live load. Column density is not changed as the loads are applied on the beams, which in turn are transferred to the columns.

Note. An alternative to modifying the material attribute as described above would be to use the section property modifier, accessed from the Attributes > Geometric > Section Property Modifier menu item, to factor the mass.

Pre-processing

To carry out a pushover study, several pre-processing steps are required:

1. Create eigenvalue structural analysis.
2. Create a structural analyses with horizontal pushover loads.
3. Specify nonlinear settings for the pushover analysis.
4. Define the nonlinear analysis controls.
5. Define and assign plastic hinges.

Eigenvalue analysis

The most common distribution of pushover forces is proportional to the first mode shape, where the lateral load f_i at storey i is given as follows:

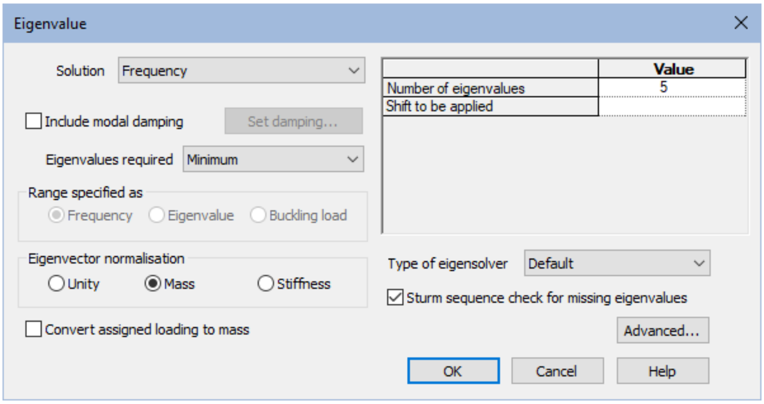
$$f_i = \varphi_i m_i \qquad \text{Eqn. 1}$$

where m_i is the mass of the i^{th} storey and φ_i is the first mode shape vector. This approach can be also found in *BS EN 1998-1:2004* Equation B.1.

Before such a load can be added, an Eigenvalue analysis must be defined.

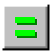
Analyses
Structural Analysis…

- On the Analysis dialog, accept the default settings but rename it to be **Modal** and click **OK**.
- Right-click on **Loadcase 2** and rename it to be **Eigen**.
- Next, right-click on **Eigen** and select **Controls > Eigenvalue…** A new window will be displayed.



- Set Solution to **Frequency**, Eigenvector normalisation to **Mass**. Enter **5** in Number of eigenvalues. Click **OK**.

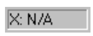
To view the eigen modes solve the analysis.

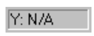
 Open the **Solve Now** dialog. Ensure **Eigenvalue** is selected and press **OK**.

The pushover load in this example will be applied only in the fundamental mode direction, so it needs to be determined if this is the global X or Y direction.

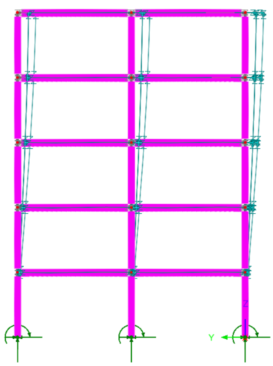
- In the Analysis  Treeview ensure the first available mode under the **Eigen** loadcase is set active.

The deformed mesh will be showing the mode shape for **Eigen mode 1.**

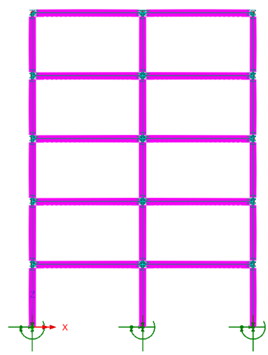
- Use the model view buttons 

and 

to view the model along relevant axes.



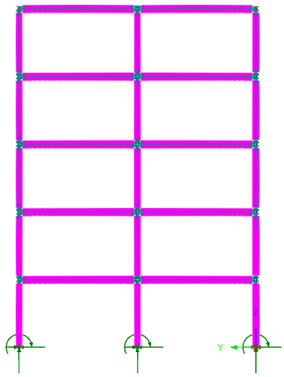
View along X-axis



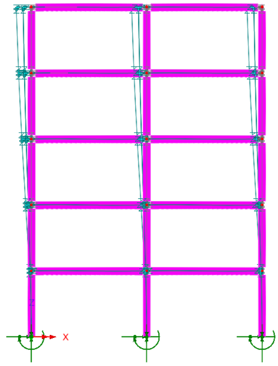
View along Y-axis

From these views it is seen that eigen mode 1 deforms the structure predominantly in Y-direction.

- In the Analysis Treeview set active **Mode 2**



View along X-axis



View along Y-axis

From these views it is seen that eigen mode 2 deforms the structure predominantly in X-direction. This is reasonable, as mode 1 deforms the columns about their minor axes, which makes it more critical. Mode 2, on the other hand, has a higher frequency, as it bends the columns about their major axes, making the response stiffer.

As a result, Mode 1 in the Y-direction will be used when referencing the pushover loads.

- In the Analysis Treeview ensure the first available mode under the **Eigen** loadcase is set active.

- Use the model view button X: N/A to view the model along the X axis.

Note. In practice, separate analyses should be carried out for each direction, but for this example only the one direction will be considered.

Pushover analysis

Create a pushover analysis.

Analyses
Structural Analysis…

On the Analysis dialog, accept the default settings but rename it to be **Pushover-Y** and click **OK**.

A pushover analysis consists of two steps. First, the full gravity loading is applied. Then the lateral load is applied incrementally to the model.

- In the Analysis Treeview rename **Loadcase 2** to be **Vertical.**

- Select **Vertical** and copy and paste it. Rename the copied loadcase to **Push-Y**.

Gravitational loading

First, apply the gravitational loading to the Vertical loadcase.

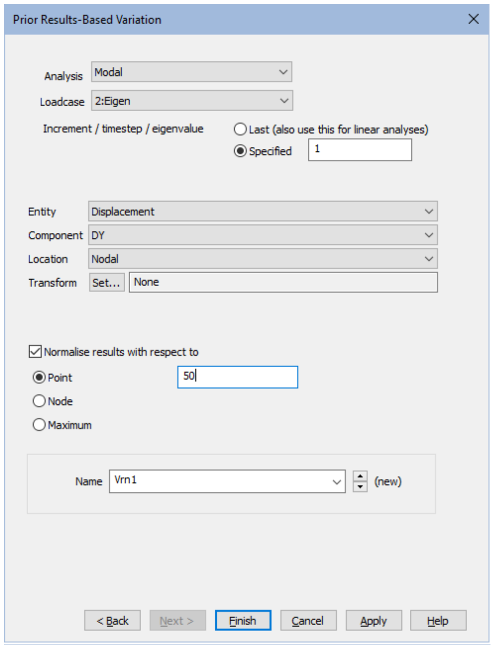
- Right-click on the loadcase **Vertical** and select **Gravity**

Pushover loading

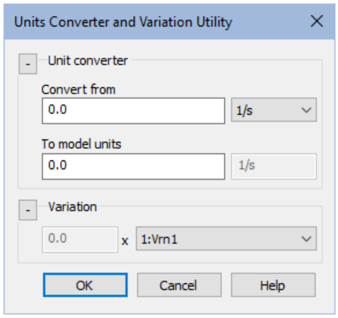
Now, define the lateral pushover loading.

Attributes
Loading…

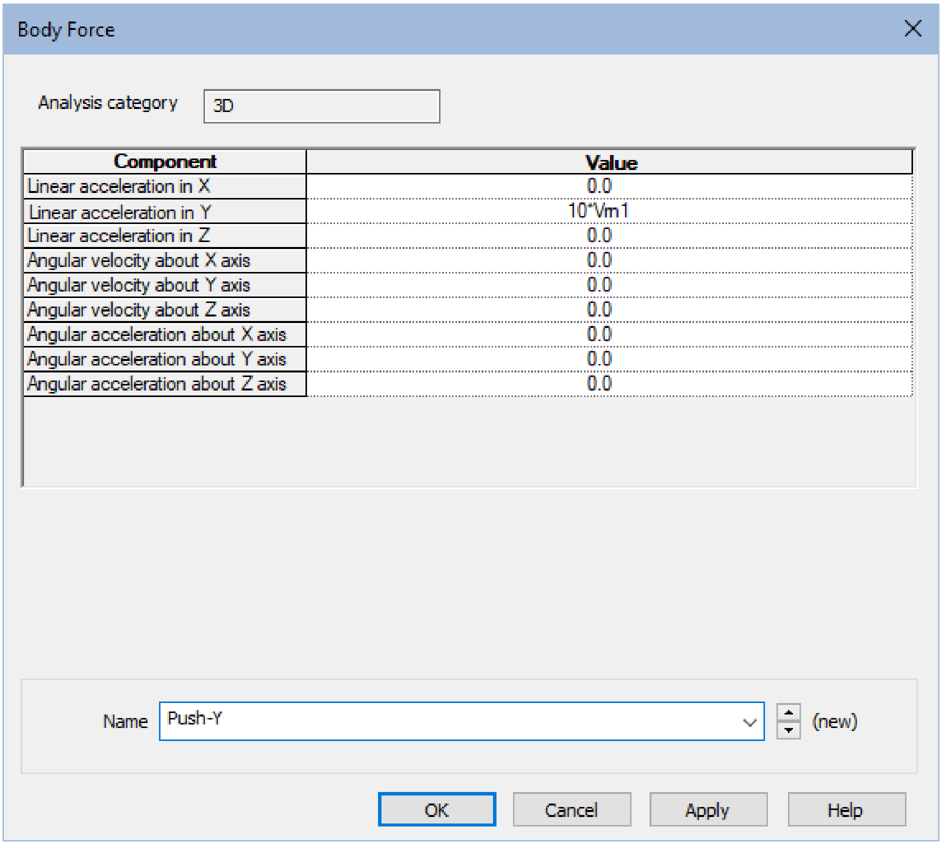
- On the Structural loading dialog, select **Body force, viscous support loading** and select **Body force**. Then press **Next**.
- For **Linear acceleration in Y**, click the arrow button. From the **Variation** droplist select **New.**
- In the new Variation window select **Prior Results-Based Variation** and press **Next**.



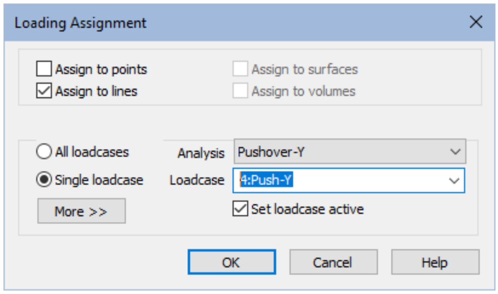
- On the Prior Results-Based Variation dialog ensure that analysis **Modal** and loadcase **Eigen** are chosen.
- Specify an increment / timestep / eigenvalue of **1**.
- Select component **DY**
- Select **Normalise results with respect to** point **50** (this is the point at the centre of the roof frame). This point will be used later on as a control feature for Pushover Curve post-processing. This normalisation ensures that a value of 1 is set here. Whilst this is not necessary, as the load will be scaled automatically, it is good practice.
- Click **Finish**.
- Back on the Variation dialog, select the newly created variation **Vrn1** in the variation droplist and click **OK**.



- Back on the Body Force dialog, for **Linear acceleration in Y,** type **10*Vrn1**. Name the attribute **Push-Y** and click **Finish**.

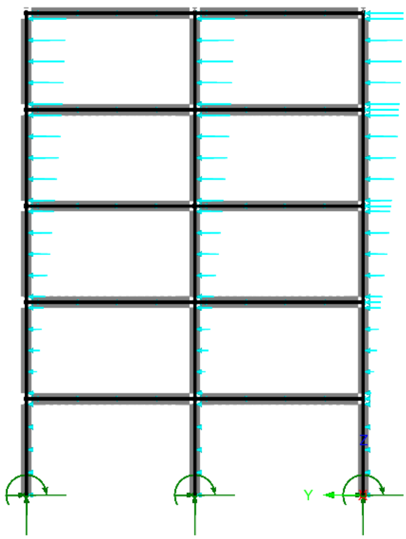


- Select all lines in the model (Press **Ctrl+A** keys). Then drag and drop the **Push-Y** load attribute onto the selection. Ensure that **Assign to lines** is only ticked. Select the loadcase name **Push-Y** and press **OK**.



All loading is now assigned to the model.

For loadcase ‘Push-Y’ the loading below will be seen.

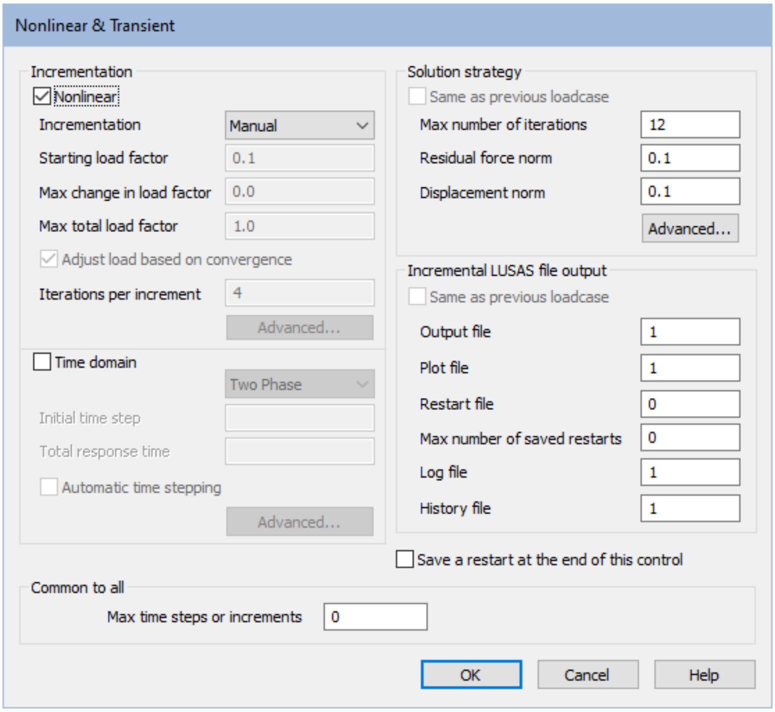


Nonlinear settings

Pushover is a highly nonlinear analysis where large displacements occur with material plastic deformations and potentially even softening. The accuracy of the analysis and convergence are controlled by nonlinear settings. Understanding them will help overcome numerical problems.

The nonlinear settings need to be specified for the Pushover-Y structural analysis.

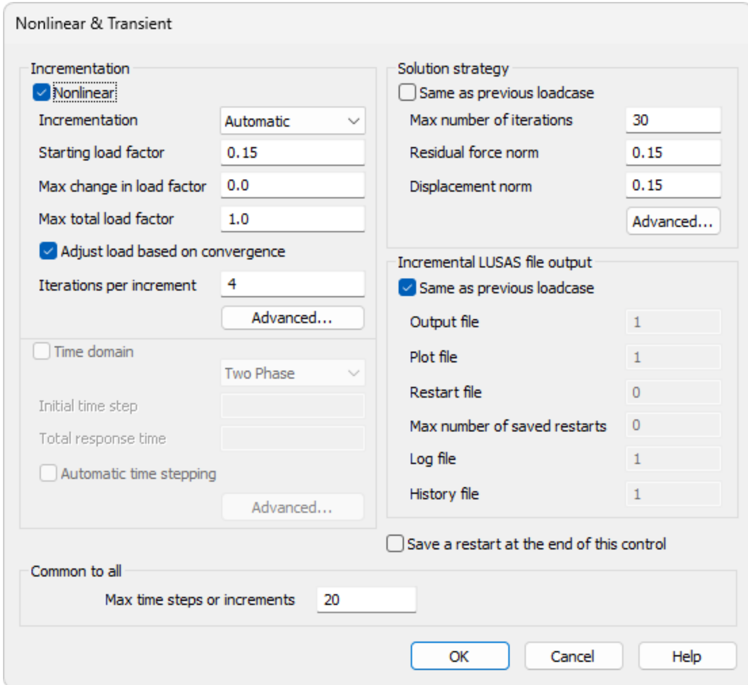
- In the Analysis Treeview right-click on loadcase **Vertical** and select **Controls > Nonlinear and Transient…**



- Tick the **Nonlinear** checkbox and ensure that the Incrementation is set to **Manual**. Press **OK**. This ensures that the full gravitational load is applied in a single step.

Now add Nonlinear & Transient controls to the 'Push-Y' loadcase in the same way.

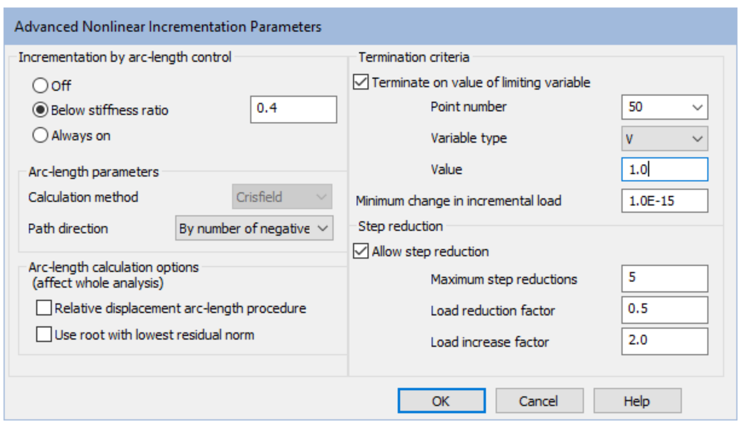
- In the Analysis  Treeview right-click on loadcase **Push-Y** and select **Controls > Nonlinear and Transient…**



- This time set Incrementation to **Automatic**, so that the loads are applied incrementally. Set **Starting load factor** to **0.15**, **Max change in load factor** to **0** and **Max total load factor** to **1.0**.

- Ensure **Adjust load based on convergence** is ticked and set **Iterations per increment** (this is referred to as '*itd'* in the Solver output file) to **4**.

- In the 'Incrementation' panel click on **Advanced…**



- On the Advanced Nonlinear Incrementation Parameters dialog, allow for automatic switching from a constant load level to an arc-length procedure by setting **Below stiffness ratio** (referred to as *cstifs* in the Solver output file) to **0.4**.

Note. In general, if an analysis is slow or fails to converge, a smaller value for the 'Below stiffness ratio' may be used or specify 'Crisfield' or 'Rheinboldt' control from the start of the analysis.

For some structures, termination criteria might be specified. It is unlikely for this structure to experience roof displacements higher than 1.0 m. Therefore:

- Tick **Terminate on value of limiting variable**, set Point number to **50**, Variable type to **V** (representing the Y-direction) with a value of **1.0**. Press **OK** on this to return to the parent dialog.

- Back on the main dialog, in the 'Solution strategy' panel, ensure that **Same as previous loadcase** is not selected, set the **Max number of iterations** to **30**, the **Residual force norm** to **0.15**, and the **Displacement norm** to **0.15**,

- In the 'Common to all' panel set the **Max time steps or increments** to **20** and then click **OK**.

Make an additional nonlinear solution setting

- In the Analysis Treeview, beneath the 'Pushover-Y' analysis entry, double-click on **Nonlinear analysis options**, and turn on **Continue solution if more than one negative pivot occurs**.

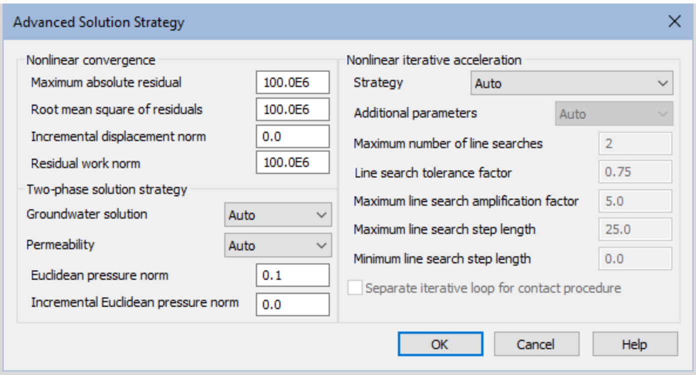
This ensures that if an analysis was to terminate because the number of negative pivots is greater than one on factoring at the start of a new increment, the solution will continue.

Optional settings (that are not required for this example)

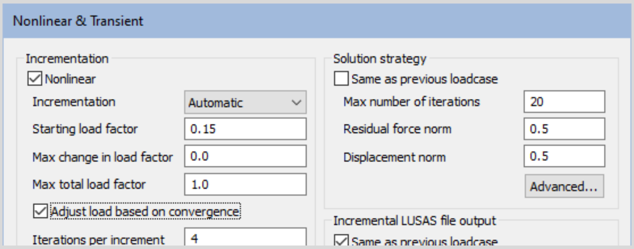
These settings are mentioned here in case of issues with running analyses on real-life projects.

To accelerate convergence

This can be done by unticking **Same as previous loadcase** on the Nonlinear & Transient dialog and pressing **Advanced…** in the **Solution strategy** groupbox. A new dialog will open. Set **Strategy** to **Auto.** Press **OK**.



Additionally, on the NL and transient dialog the convergence criteria can be relaxed by setting **Residual force norm** and **Displacement norm** (found on the parent Nonlinear & Transient dialog) to **0.5**. This can make the analysis faster but reduces the accuracy. If a softening response is required, these parameters should not be increased.

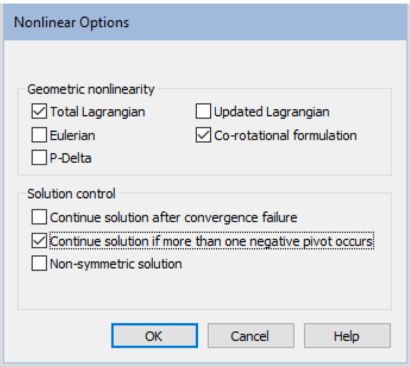


To account for geometric nonlinearity

In addition to setting Nonlinear Controls, the use of some additional model option settings is also recommended.

If geometric nonlinearity is required, it can be included by checking the **Total Lagrangian** checkbox on the Nonlinear options dialog. This is accessed from the Nonlinear Options entry in the Analyses treeview, by double-clicking on **Nonlinear analysis options** under 'Pushover-Y' analysis.

For beam element models, the **Co-rotational formulation** option should also be chosen.



To overcome numerical problems

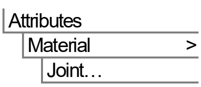
As softening is modelled in the hinge, numerical problems can occur. To attain structural response beyond this point, **Residual force** norm and **Displacement norm** values (that are set in the **Solution strategy** settings of theNonlinear & Transient dialog) should be reduced e.g. to **0.1**. This may require a longer analysis time and the creation of more increments.

Defining plastic hinges

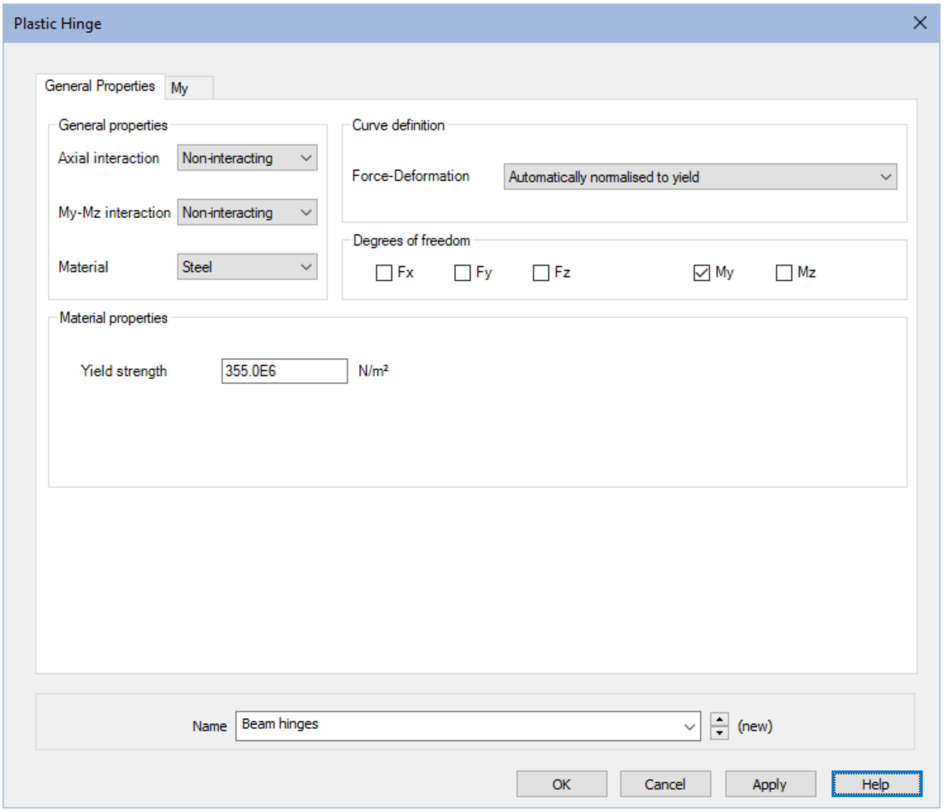
Plastic hinges for beams and columns now need to be defined. Beams do not experience much axial force, so simple non-interacting hinges can be used to define them. Columns, on the other hand, are subjected to substantial axial forces, which can impact their capacity in bending. Therefore, columns require axial interaction. If biaxial bending is expected, My-Mz interaction is needed as well.

Beam hinges

First, a simple beam hinge is defined.



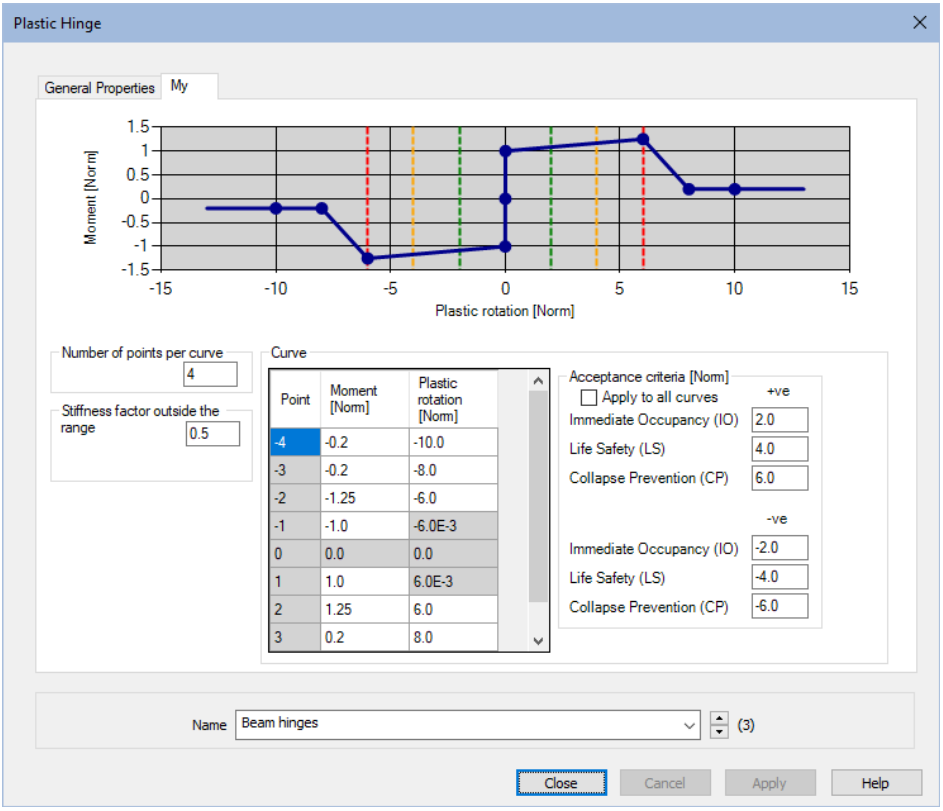
- On the dialog select **Plastic Hinge** and click **Next**.



- On the General Properties tab, in the **Axial interaction** droplist select **Non-interacting**.
- Since only bending about the major axis is expected, for **My-Mz interaction** select **Non-interacting**.
- For **Material** select **Steel**. The curve definition Force-Deformation droplist will be set to **Automatically normalised to yield**, automatically calculates the force deformation curves for each hinge based on the assigned attributes and the specified yield stress. In this way the same hinge attribute with a normalised curve definition can be dropped on elements with different section (geometric attributes) or length.
- For **Degrees of freedom**, ensure that only **My** is ticked.
- For **Yield strength** leave the default value of **355E6** N/m^2 set, which reflects the use of S355 steel.
- Name the attribute **Beam hinges** and proceed with more settings…

My tab

- Next, select the **My** tab to define a force-deformation curve (bending moment-rotation curve).



- Leave the default settings as they are, noting the following:

Note. In the curve definition, the values of moment and plastic rotation are entered as normalised values and will be scaled by the calculated yield values. For instance, bending moment at yield is given by:

$$M = ZF_y \qquad \text{Eqn. 2}$$

Whilst rotation at yield is given as:

$$\Theta_y = \frac{ZF_yL}{6EI} \qquad \text{Eqn. 3}$$

Equation 3 is based on FEMA 356 Equation 5-1.

Note. The default curve definition could be used for generic steel materials, but you are encouraged to modify them for projects as needed. Remember that the displacements must be monotonically increasing. Acceptance criteria can be modified, as prescribed by the specific code of practice e.g. ASCE 41-17. As these values are changed, the shape of the diagram at the top of the dialog is updated. The acceptance criteria are shown with dashed vertical lines: green for IO, orange for LS and red for CP.

The number of points per curve is set to 4.

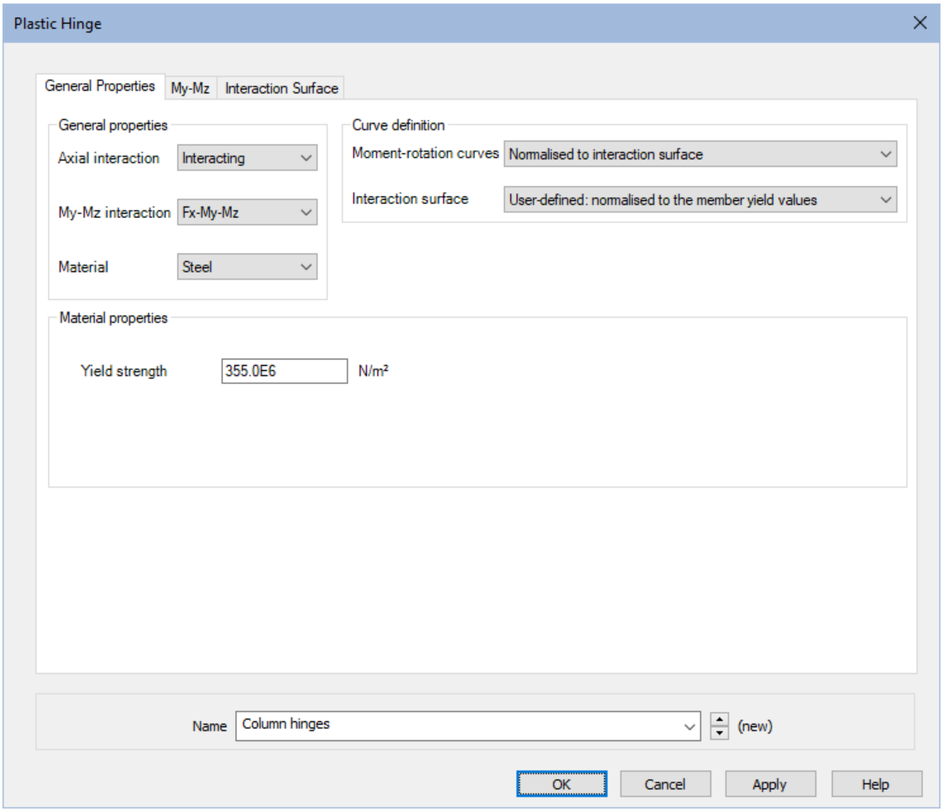
- Point 1 indicates the yield point.
- Point 2 shows the ultimate strength. The segment between points 2 and 3 is the softening range. It is recommended to make it no steeper than 10% of the hardening portion, as sudden loss of strength can cause convergence issues.
- Points 3 and 4 indicate the residual strength.

Note. The value of **Stiffness factor outside the range**, changes the slope of the curve beyond the user-defined displacements and can take a value from 0 to 1. Since the slope of the final segment (between points 3 and 4, as well as -3 and -3) is zero, this factor plays no role. However, if the slope of the final segment is non-zero, the resultant slope is indicated in the diagram by the extension segments on each end. If the extension reaches the X-axis, the residual force/moment of zero is maintained.

- Click **Apply** to save the Beam hinges attribute, and define another attribute.

Column hinges

- With the Plastic Hinge dialog still displayed, in the **Name** box, replace ‘Beam hinges’ by over-typing the name to be **Column hinges**



- On the **General Properties** tab change **Axial interaction** to **Interacting**. Although bi-axial loading is not applied in this example, ensure **Fx-My-Mz** interaction is selected under **My-Mz interaction**, to illustrate how they can be defined.

Note. Whilst biaxial interaction is not expected in the current analysis, defining the hinge with interaction will allow it to be used in multiple pushover analyses considering different load directions.

- Set the **Moment-rotation curves** droplist to **Normalised to interaction surface**. This will calculate section yield values accounting for the specified interaction surface.

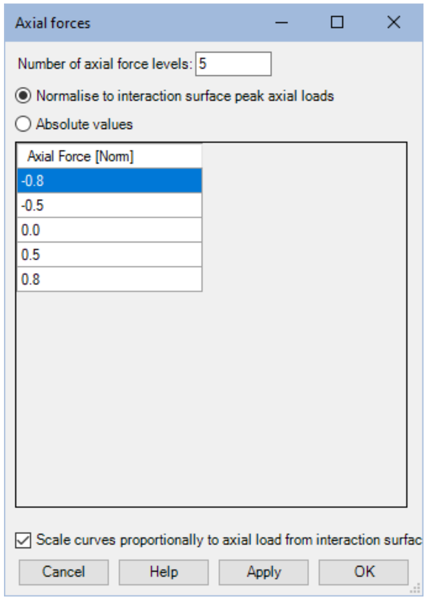
- Set the **Interaction surface** to **User defined: normalised to the member yield values**. You will need to define the interaction surface in terms of normalised bending/axial compression yield values.

My-Mz tab

- Now select the **My-Mz** tab to define the moment-rotation curves.

The number of curves defining the joint is controlled by the number of axial force levels and angles.

- Click on the **Axial forces** button to open its dialog.



- Leave the number of axial forces levels set to **5**.

- Ensure **Normalise to interaction surface peak axial loads** is selected. This means that the hinge attribute can be defined for any section in terms of its axial capacity. The true axial force level $F_{x,n}$ for an entry n is calculated as follows:

$$F_{x,n} = F_{x,n,norm} \times F_{x,y} \qquad \text{Eqn. 4}$$

where $F_{x,y}$ is the yield axial force of the section with area A and yield strength f_y calculated as follows:

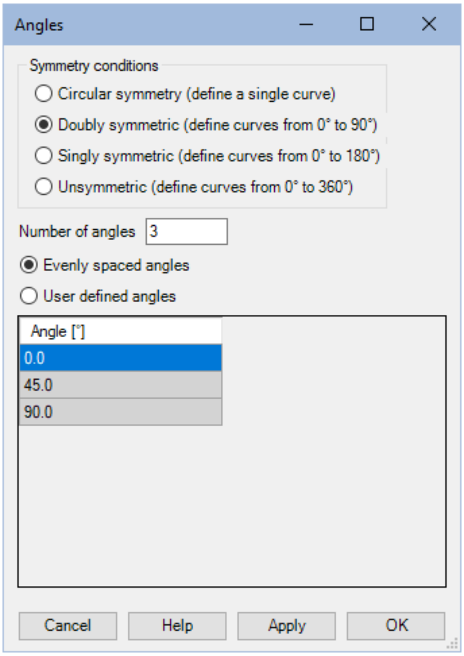
$$F_{x,y} = Af_y \qquad \text{Eqn. 5}$$

- Define Axial Force [Norm] values of **-0.8**, **-0.5**, **0**, **0.5** and **0.8** as per the dialog shown above.

- Ensure **Scale curves proportionally to axial load from interaction surface** is ticked. This means that only a single curve at each angle needs to be defined for $F_x = 0$. Other curves are scaled proportionally by the normalised axial force ratio. For example, for an axial force equal to half the axial capacity (such as *0.5 Norm* or *-0.5 Norm*), the curve moments and rotations are reduced by half.

- Click **OK** to close the dialog.

- Back on the Plastic Hinge dialog, click on the **Angles** button to open its dialog.



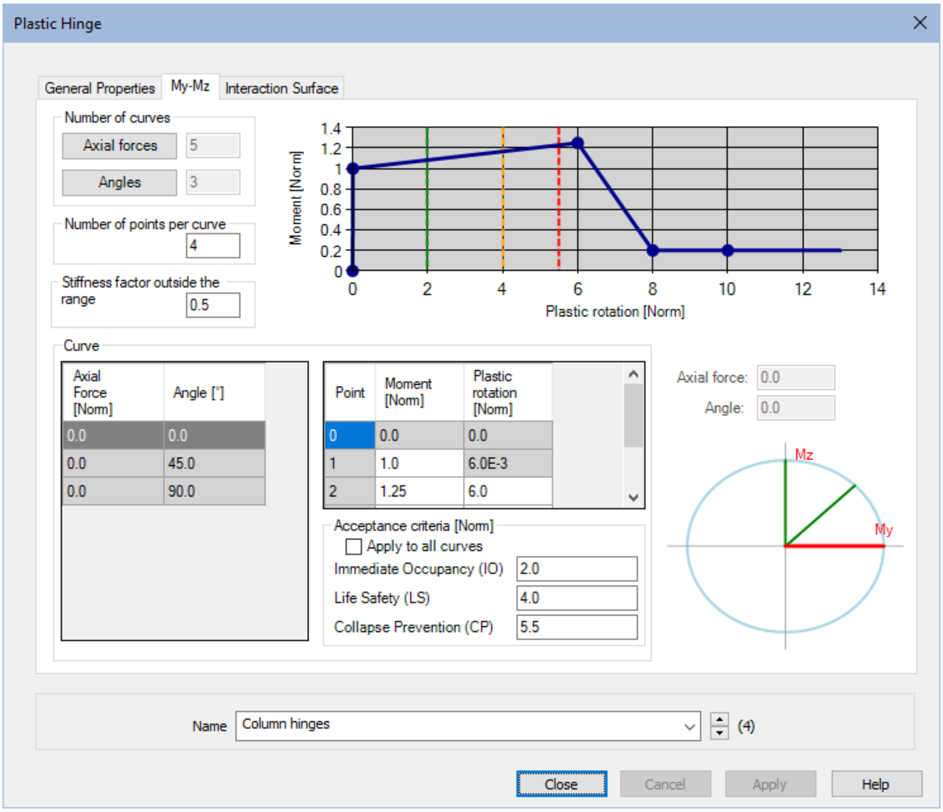
- Ensure that the **Doubly symmetric** radio button is selected. Since an I-section is doubly symmetric about major and minor axes the curves from 0º to 90º only need to be defined.

- Change the number of angles to **3**. (Click elsewhere on the dialog to ensure this value is taken. The grid will update when done).

- Select **Evenly spaced angles**, For this the program automatically selects the angles at equal intervals. Click **OK**.

Back on the Plastic Hinger dialog, because the option 'Scale curves proportionally to axial load from interaction surface' was selected on the Axial forces dialog, the number of curves that need to be defined is now three:

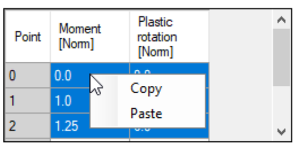
- ❑ 0 degrees at 0 kN axial load.
- ❑ 45 degrees o at 0 kN axial load.
- ❑ 90 degrees at 0 kN axial load.



Outside of this example, if a curve needs to be modified, choose the corresponding entry in the Axial Force / Angle grid. As you change the curves, both diagrams are updated accordingly.

The Axial Force / Angle diagram in the bottom-right corner of the dialog shows the curve angles drawn in green lines. The currently selected angle is shown in red. Additionally, the current axial force and angle are shown in the greyed out boxes above this diagram.

Each curve can be defined in the grid in the centre of the dialog. Note that if you want to copy or paste these values to/from Excel, you can select the desired cells, right-click on them where a context menu with **Copy** and **Paste** options will appear.



- The **Collapse prevention (CP)** limit at normalised plastic rotation of 6 might be a bit too tight, so change this to be **5.5** to give some leeway for small deviations.

- Then tick **Apply to all curves**, which will lock-in the Acceptance Criteria values for all curves.

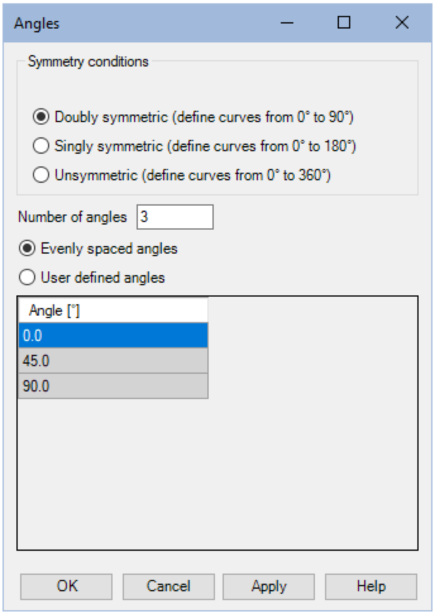
Interaction surface tab

- Select the **Interaction Surface** tab.

The layout of the dialog is similar to that for force-deformation curves. First the number of angles needs to be defined.

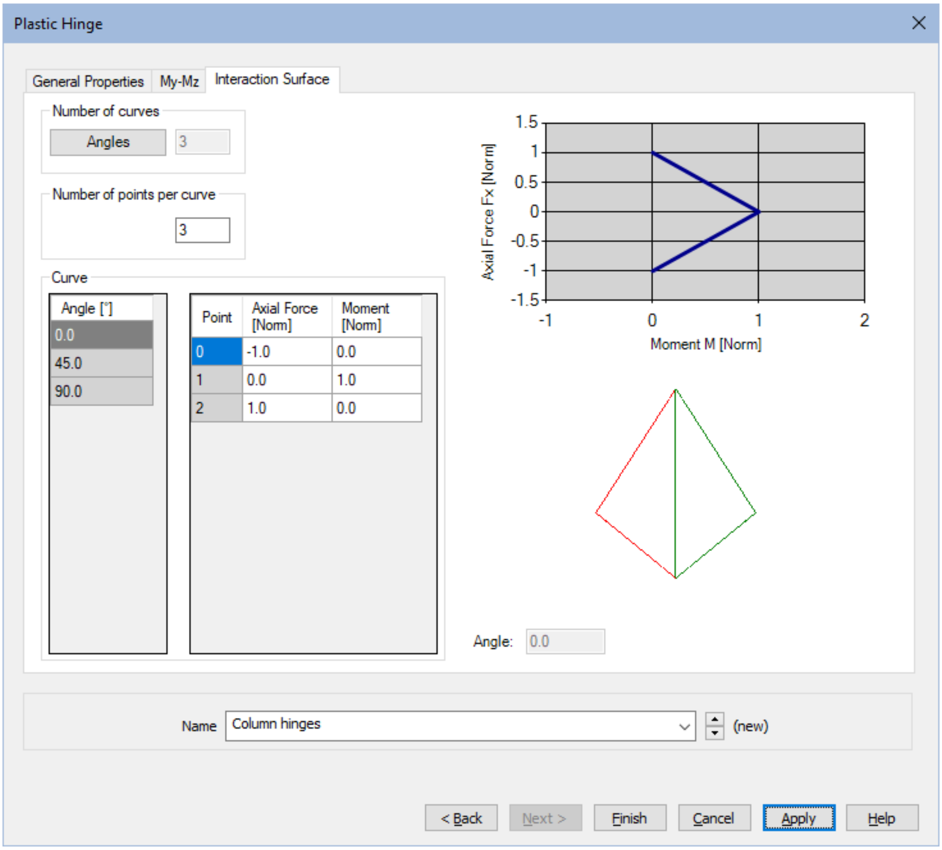
- Click on the **Angles** button to open its dialog

It is recommended that the angles in the interaction surface and moment-rotation tabs are the same. Therefore, the settings should be as shown below.



- Click **OK** to close this dialog.

Back on the main Plastic Hinge dialog:



Just as for the force-deformation tab, the interaction curves are shown in green in the bottom-right 3D interacting diagram, with the currently selected entry shown in red. The diagram in the top right corner shows the currently selected interaction curve detailed definition. Note that it is posible to rotate it or zoom in and out of the interacting diagram.

- Press **Finish** to save the attribute and close the dialog.

Note. The default interaction curve values employ a linearly-varying surface, which provides a simple defintion. The axial force and moments are normalised to their yield capacities. For axial force, refer to Equation 5. The yield moment $M_{\alpha,y}$ at an angle α is given as:

$$M_{\alpha,y} = M_{y,y} \times \cos(\alpha) + M_{z,y} \times \sin(\alpha) \qquad \text{Eqn. 6}$$

where $M_{y,y}$ and $M_{z,y}$ are the yield moments about the major and minor axis respectively calculated from Equation 2.

Similarly, the yield rotation $\theta_{\alpha,y}$ at angle α is given as:

$$\theta_{\alpha,y} = \theta_{x,y} \times cos(\alpha) + \theta_{y,y} \times sin(\alpha) \qquad \text{Eqn. 7}$$

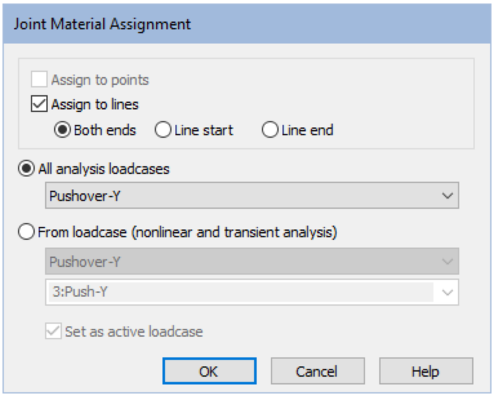
where $\theta_{y,y}$ and $\theta_{z,y}$ are the yield rotations about the major and minor axis respectively calculated from the following equation:.

$$\theta_y = Z \times f_y \times L/(6EI) \times (F_x/F_{x,y}) \qquad \text{Eqn. 8}$$

Where Fx is the current axial force and Fx,y is the yield capacity from Equation 5. Note that as opposed to Equation 3, Equation 8 accounts for the influence of axial force in the member. This is based on Equation 5-2 for columns from FEMA 356.

Assign the beam and column hinges

- With no features selected, in the Attributes treeview, right click on the **Beam mesh** attribute and choose **Select Assignments**, then drag and drop the **Beam hinges** attribute into the selected features, assigning to **Both ends** of the beams, and noting that springs should not be applied in an Eigenvalue analysis, so in **All analysis loadcases** select **Pushover-Y** instead, as shown below. And click **OK**.

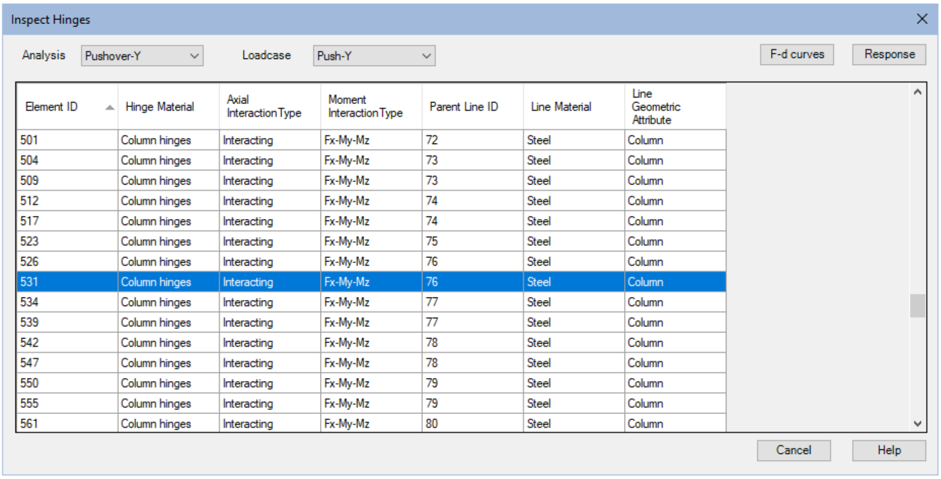


- Right click on the **Column mesh** attribute and choose **Select Assignments**, click **OK** to confirm clearing the previous selection, then drag and drop the **Column hinges** attribute into the selected features, assigning to **Both ends** of the columns, and ensuring that they are assigned to the **Pushover-Y** analysis also and click **OK**.

True force-deformation curves

Tools
Inspect Hinges…

This facility allows you validate the plastic hinges and view the true force-deformation curves for a particular element.



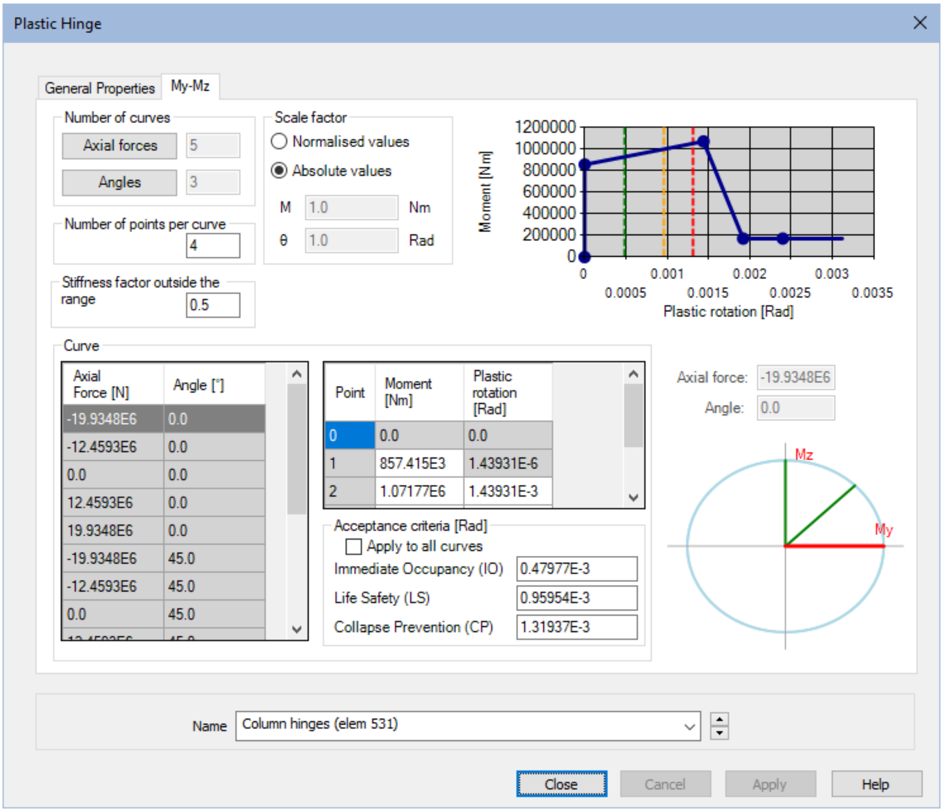
Element ID	Hinge Material	Axial Interaction Type	Moment Interaction Type	Parent Line ID	Line Material	Line Geometric Attribute
501	Column hinges	Interacting	Fx-My-Mz	72	Steel	Column
504	Column hinges	Interacting	Fx-My-Mz	73	Steel	Column
509	Column hinges	Interacting	Fx-My-Mz	73	Steel	Column
512	Column hinges	Interacting	Fx-My-Mz	74	Steel	Column
517	Column hinges	Interacting	Fx-My-Mz	74	Steel	Column
523	Column hinges	Interacting	Fx-My-Mz	75	Steel	Column
526	Column hinges	Interacting	Fx-My-Mz	76	Steel	Column
531	Column hinges	Interacting	Fx-My-Mz	76	Steel	Column
534	Column hinges	Interacting	Fx-My-Mz	77	Steel	Column
539	Column hinges	Interacting	Fx-My-Mz	77	Steel	Column
542	Column hinges	Interacting	Fx-My-Mz	78	Steel	Column
547	Column hinges	Interacting	Fx-My-Mz	78	Steel	Column
550	Column hinges	Interacting	Fx-My-Mz	79	Steel	Column
555	Column hinges	Interacting	Fx-My-Mz	79	Steel	Column
561	Column hinges	Interacting	Fx-My-Mz	80	Steel	Column

- Select **Pushover-Y** in the Analysis droplist and **Push-Y** from the Loadcase droplist.. This will list all the plastic hinge elements in the given loadcase.
- Click the **Element ID** column header to sort the entries numerically.
- Select a row with Hinge material '**Column Hinges**' (an example of element ID **531** is selected here) and press the **F-d curves** button.

This opens the **Plastic Hinge** definition dialog for the chosen element, but note that the attribute is converted to be applicable for **Any** material model, so that true force-deformation curves with absolute values can be inspected.

Note. The generated attribute name of "Original attribute name (elem <number>)". does not exist explicitly in the model, but pressing 'OK' or 'Apply' would add it to the model. There is no need to do it for this study.

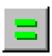
- Select the **My-Mz** tab. It can be seen now that 15 curve definitions are available: five for each axial force level and three for each angle. The curves are tabulated using absolute values. This allows inspection of absolute values used in the analysis and shows that defining and viewing details of plastic hinges is easily achieved.



- Click **Close** on this dialog and **Cancel** on its parent.

Modelling is now complete. The pushover analysis can be solved.

Running the Analysis

 Open the **Solve Now** dialog. Ensure analysis **Pushover-Y** is selected and press **OK**.

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

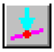
As this is a Nonlinear analysis, it might take a few minutes to solve.

Post-processing

This section describes how to post-process the results to determine the performance point in the analysed model and investigate the distribution of hinge formation across the structure.

First:

 Turn off the Fleshing.

 Turn off the Loading visualisation.

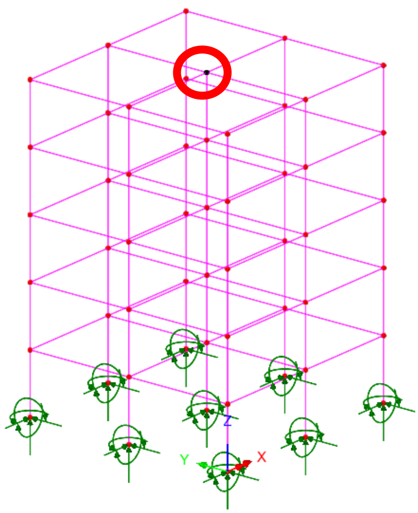
- In the Layers  Treeview ensure the mesh and deformed mesh layers are turned off.

Pushover curve

The response of the structure is investigated by extracting the pushover curve.

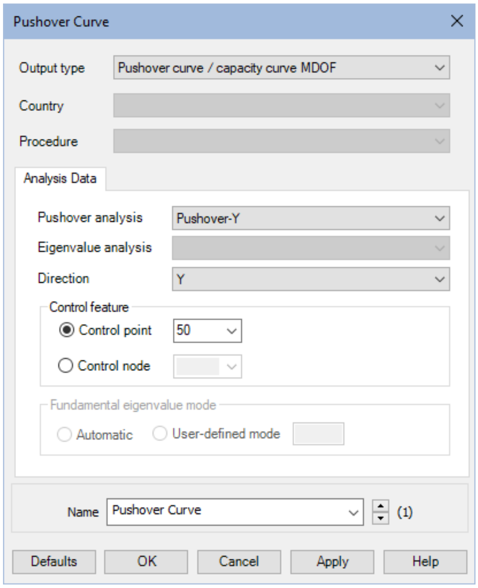
Firstly, a **control feature** needs to be selected. Typically, a point in the top floor is used, so a point in the centre of mass, or nearby is a suitable choice.

- Rotate the model to give a similar view to that shown below.
- With the geometry layer displayed, hold down the **P** key (to select a Point) and drag a selection box around the point in the top floor as shown. Point **50** should be selected.



Utilities
Pushover Curve…

On the pushover curve dialog the selected point is automatically loaded as the control feature. If several points/nodes were selected, the droplist would list them all (up to 100).



- To obtain a pushover curve, also refered to as a ‘Capacity curve MDOF’ in the Eurocode, select **Pushover curve / capacity curve MDOF** in the **Output type**.

There is only one valid Pushover analysis, **Pushover-Y**, which will be already selected.

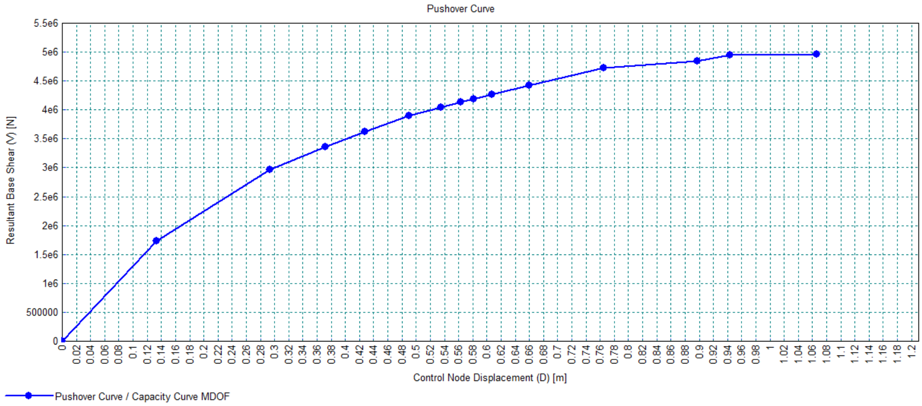
Note. A valid pushover analysis must include Nonlinear & transient controls for at least two loadcases: the first defines the vertical load (using manual incrementation) and the other defines the lateral load application (using automatic incrementation).

- Select **Y** for the direction of the lateral loads in the Y-direction

- Rename the attribute to be **Pushover Curve** and click **OK**.

The pushover curve is displayed in the graph. This plots the displacements in the Y-direction of the control points on the X-axis vs base shear on the Y-Axis. Base shear is total load applied on the structure at the given step. The number of points on the graph is equal to the number of increments in the given analysis. It is assumed that the vertical load application in the first loadcase determines the starting point.

The graph shows a typical building response: an initially elastic response, followed by the spread of hinges and flatter plastic behaviour, concluding with some softening after which a failure/collapse can be assumed.



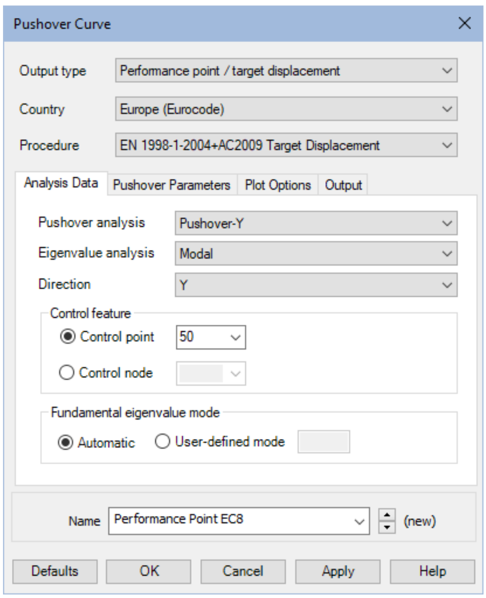
- Go back to the model view.

Determine target displacement

LUSAS can process a pushover curve to determine the 'performance point' (USA) or 'target displacement' (Europe) according to building codes. This example illustrates how to find the target displacement according to *EN 1998-1-2004*. To learn more about this procedure refer to *Fajar* (2021) showing how the N2 method was developed, and which was incorporated into Eurocode in a slightly modified form.

Utilities
Pushover Curve…

Open the pushover curve dialog again.

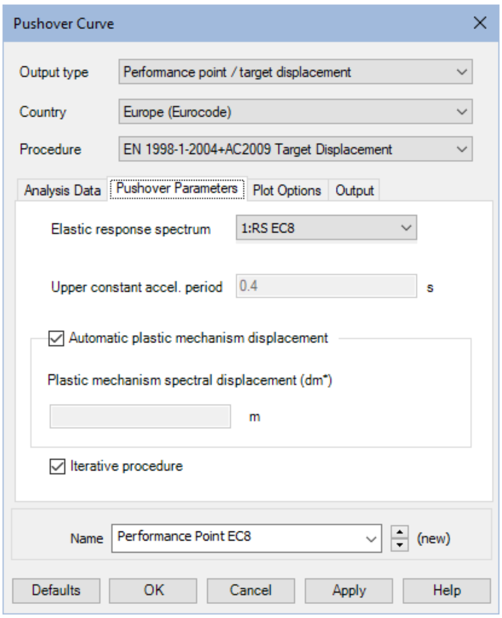


Analysis data

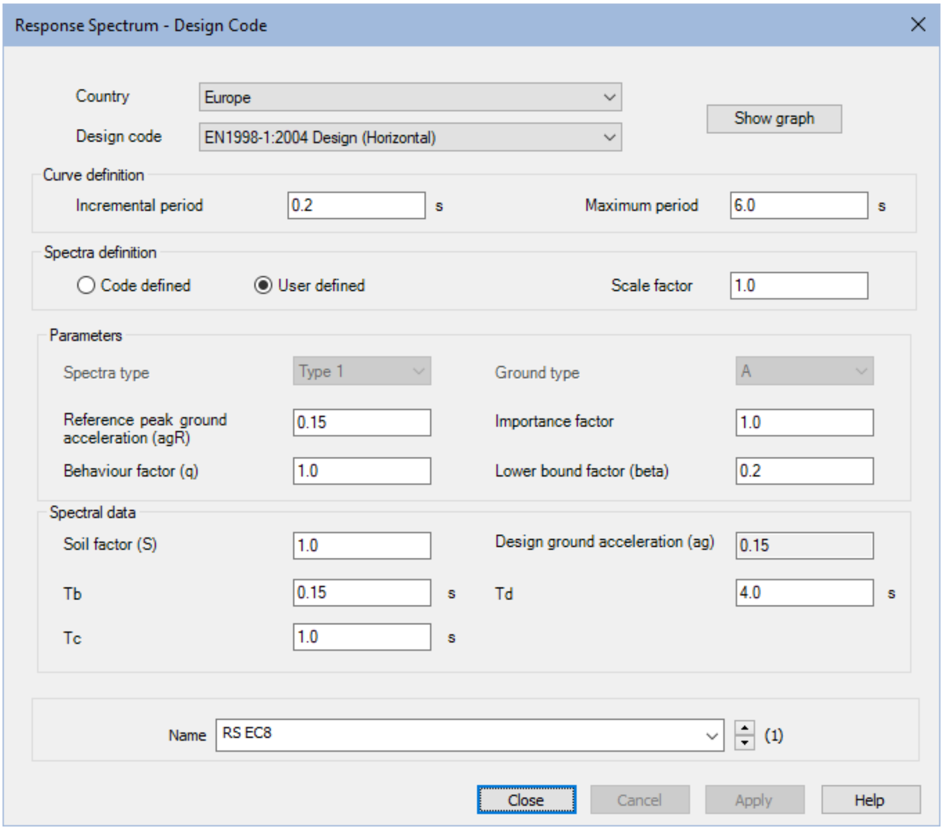
- First, rename the utility to **Performance Point EC8**.
- For Output type select **Performance point / target displacement**. Select country **Europe (Eurocode)**. For Procedure, select **EN 1998-1-2004+AC2009 Target Displacement**
- For **Pushover analysis** select **Pushover-Y** (as used in the previous step). The eigenvalue analysis droplist will list all structural analyses with eigenvalue controls. **Modal** is the only valid analysis, which is already selected.
- Select **Y** for the direction of the lateral loads is in the Y-direction
- Ensure that control point **50** is entered.
- The facility can determine automatically the fundamental eigenvalue mode for the given direction. But since this has already been determined to be mode 1, select **User-defined mode** radio button and enter **1**.

Pushover parameters

- Select the **Pushover Parameters** tab to define procedure-specific settings.



- A Eurocode elastic response spectrum has already been defined in this model. To view it, open the **Elastic response spectrum** droplist and next to **RS EC8** click **Edit…**.



Code based values are initially displayed, but for this example the spectra definition will be user-defined.

- Select **User-defined**
- In the Parameters panel enter a **Reference peak ground acceleration (agR)** of **0.15**
- In the Spectral data enter **Td** to be **4.0** and enter **Tc** to be **1.0**
- Ensure all other settings are as per the dialog and click **OK** to close it.

In practice these parameters should reflect the seismic conditions of the site where the structure will be built.

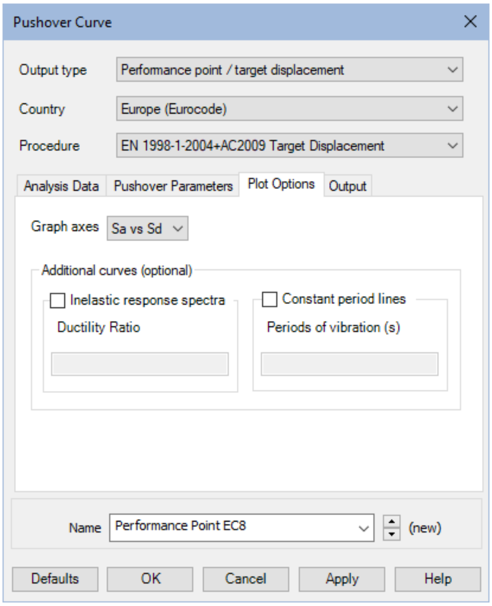
Back on the Pushover Curve dialog and its Pushover parameters tab, since the response spectrum is defined, the **Upper constant accel. period** is greyed out. In Eurocode this is denoted as T_C.

The starting plastic mechanism spectral displacement (d_m^*) can be user-defined. But as a starting point it is recommended to keep the **Automatic plastic mechanism displacement** checked. This is taken as the point at the peak spectral acceleration.

EC8 Section B.5 allows the use of an optional iterative procedure. It is highly recommended to tick **Iterative procedure** checkbox to get more accurate results. Without iteration the target displacements may be grossly overestimated, resulting in an over-conservative design.

Plot Options

From the **Plot Options** tab, you can control the graph axes.

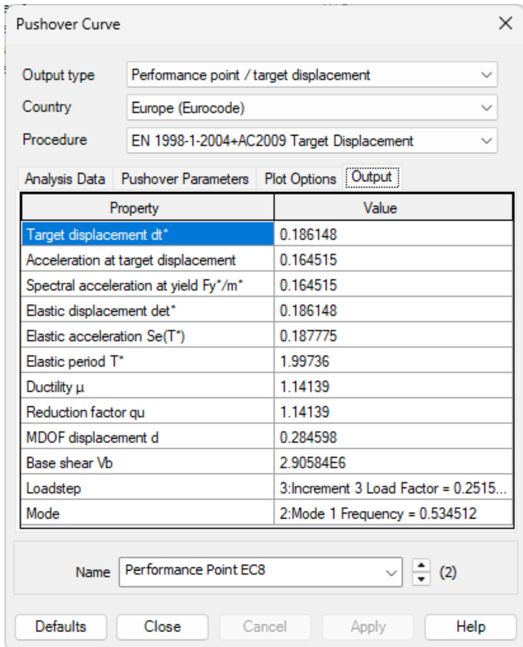


- Ensure that the graph axes results are drawn in **Sa vs Sd** format, which is a typical selection.

Note. You may optionally plot user-defined inelastic response spectra or constant period lines. These can be used to plot the elastic spectrum, if the ductility μ is set to 1. However, these options are left unticked for this worked example.

- Click **Apply** to determine the performance point.

Output

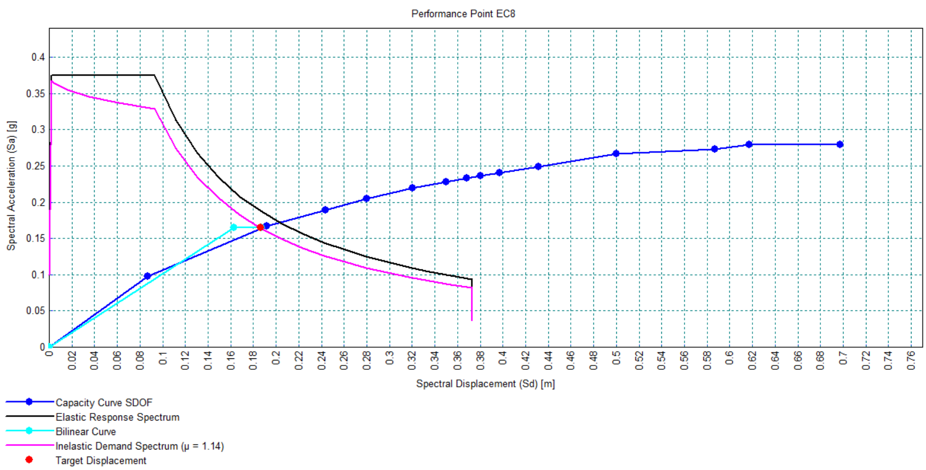


The results will be drawn in the graph behind the dialog, but before closing the dialog, note that the key results are tabulated in the **Output** tab. These include:

- ❑ Data about the performance point.
- ❑ The target displacement in spectral format (SDOF), as well as the actual MDOF reponse in the structure.
- ❑ The increment in which the displacement is found is tabulated alongide the **Loadstep** property.

For the settings made, the performance point is found to occur at Loadstep increment 4. This information will be needed later when investigating the structure.

- Close the dialog to inspect the graph.



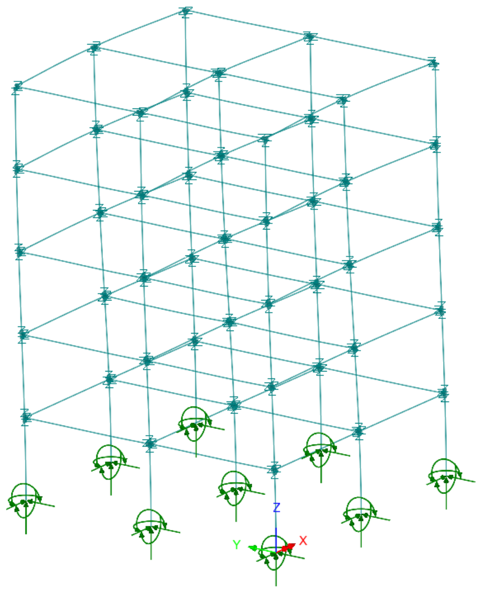
On the graph the blue curve represents the *capacity curve* i.e. the *pushover curve* in SDOF spectral format. The intersection of the *inelastic demand spectrum* with *capacity* *curve* determines the *target displacement / performance point*, which is shown by a red dot.

- Close the graph to return to the main model. It can be re-displayed, if needed, by right-clicking on the **Pushover Curve** name and choosing **Display graph**.

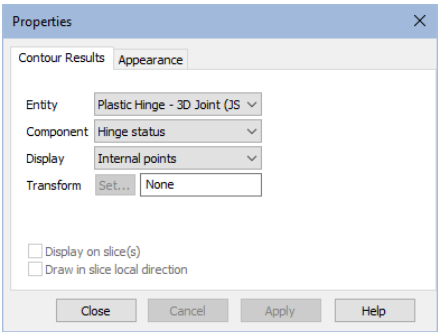
Hinge status for maximum load factor

- In the Analysis Treeview set active the **last** loading increment in order to then view the entire spread of plastic hinges in the structure.
- Turn on the **Deformed mesh** layer.
- Turn off the **Geometry** layer

The deformed shape will be seen, but of more interest is the status of the hinges.

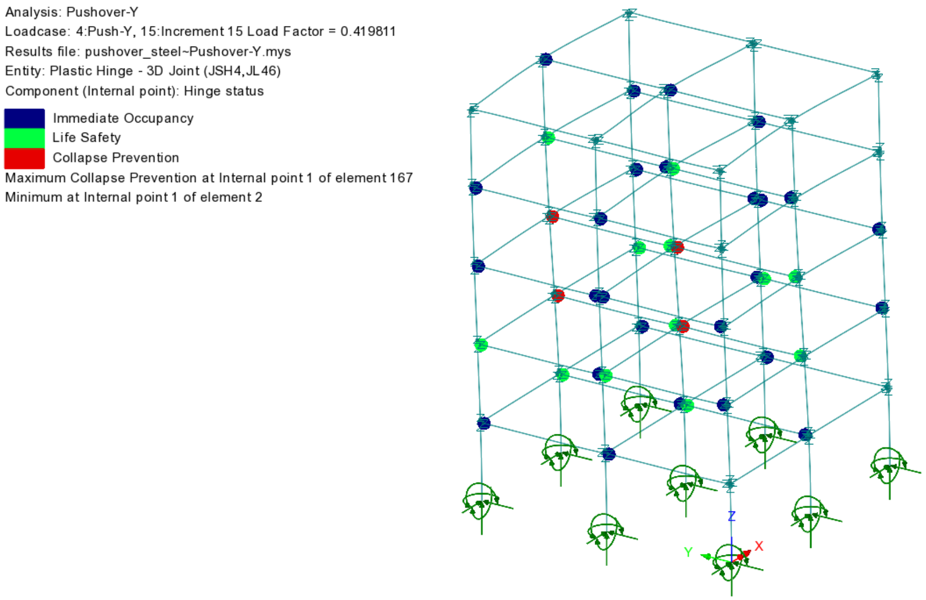


- In the model view window, right-click in a blank region and selected **Contours**. Select entity **Plastic Hinge – 3D Joint (JSH4,JL46)**, component **Hinge status**



- Select the **Appearance** tab and press **Set..** under **Classic** radiobutton. Set **Width** to **15** to make sure the coloured blobs that will be drawn at the locations of hinges are sufficiently large.
- Click **OK** and **OK** again to close the dialogs and add the contours layer to the Layers Treeview.

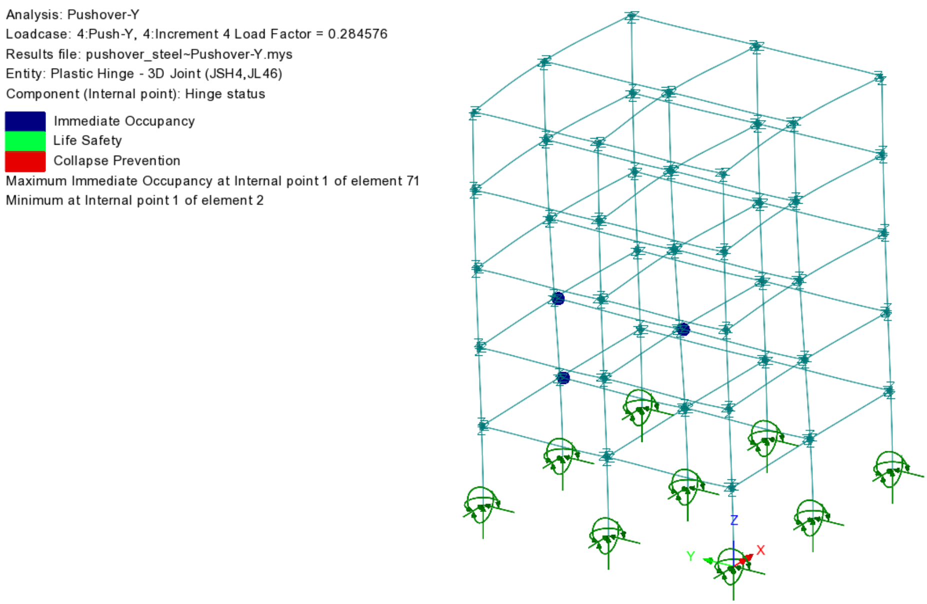
For the last loading increment, the following plot will be seen.



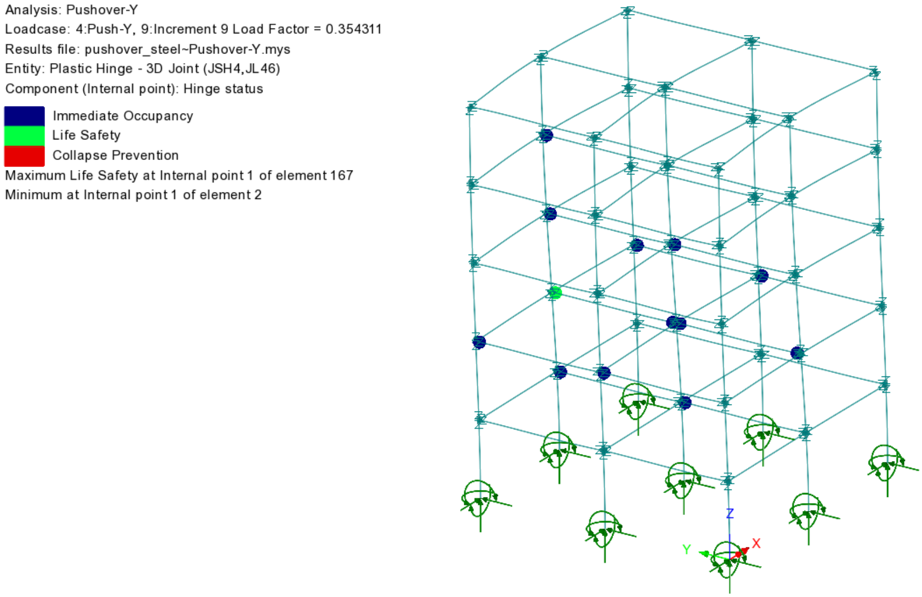
From the contour blobs showing plastic hinge status, it can be seen that Immediate Occupancy status (blue) has been reached in many joints, Life Safety status (green) in 12 joints, and Collapse Prevention status (red) has been reached in 4 joints – one of which is element 167.

Hinge status for selected load increments

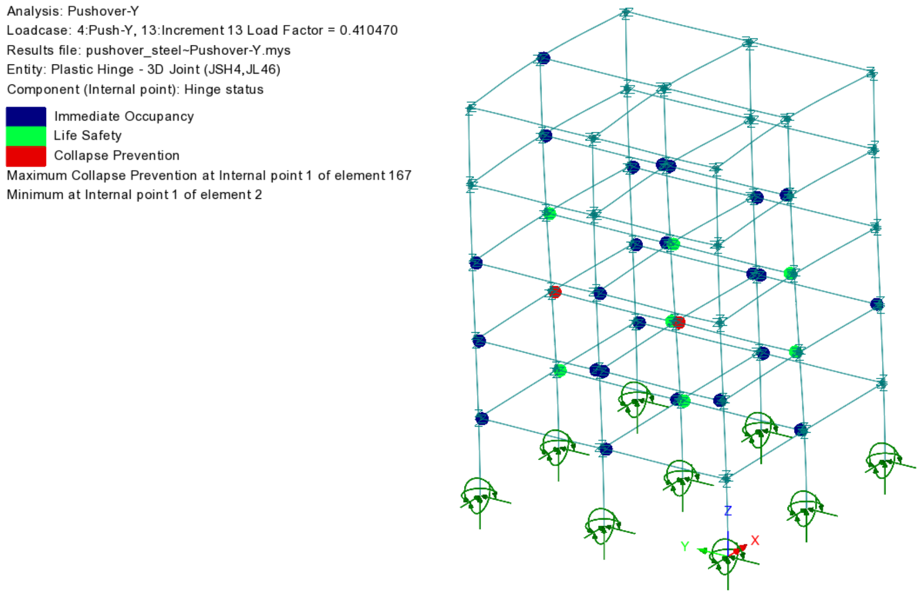
- In the Analysis Treeview, set active **load increment 4** to view the plastic hinge status at the target displacement / performance point. Three hinges can be seen to have reached Immediate Occupancy status (blue) at a load factor of around 0.285



- By setting active load increments in turn, it can be seen that Life Safety status (green) is reached first at **load increment 9** and also in element 167 with a load factor of 0.354

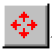


- It can be also be seen that Collapse Prevention status (red) is reached at **load increment 13** with a load factor of 0.410



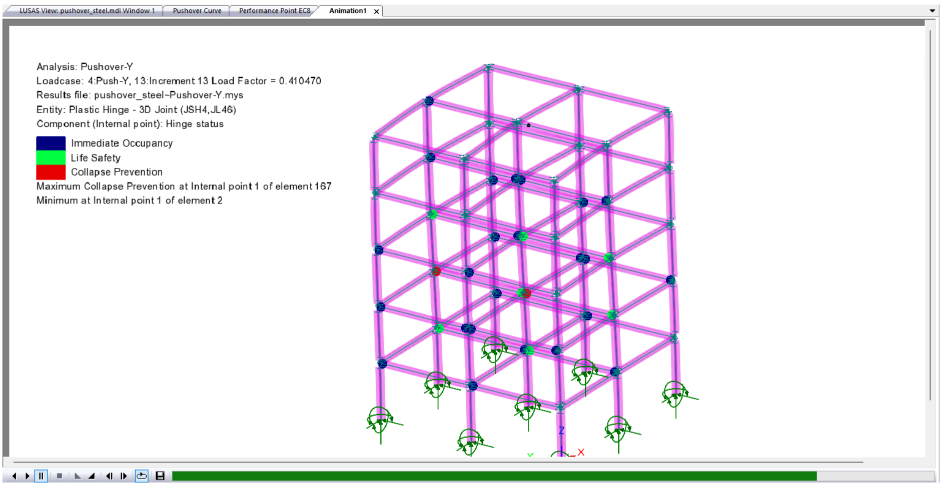
Animating the results

The spread of plastic hinges through the structure may be appreciated more easily by creating an animation of the load increments applied. In preparation for this:

- In the toolbar, press  followed by  to view the structural members with transparency.
- At the bottom of the Layers  Treeview panel, click the **Deformations** button and set the factor to **1.0**
- To ensure that the model is not re-sized within the view window for each increment, press to turn off the resize ability .

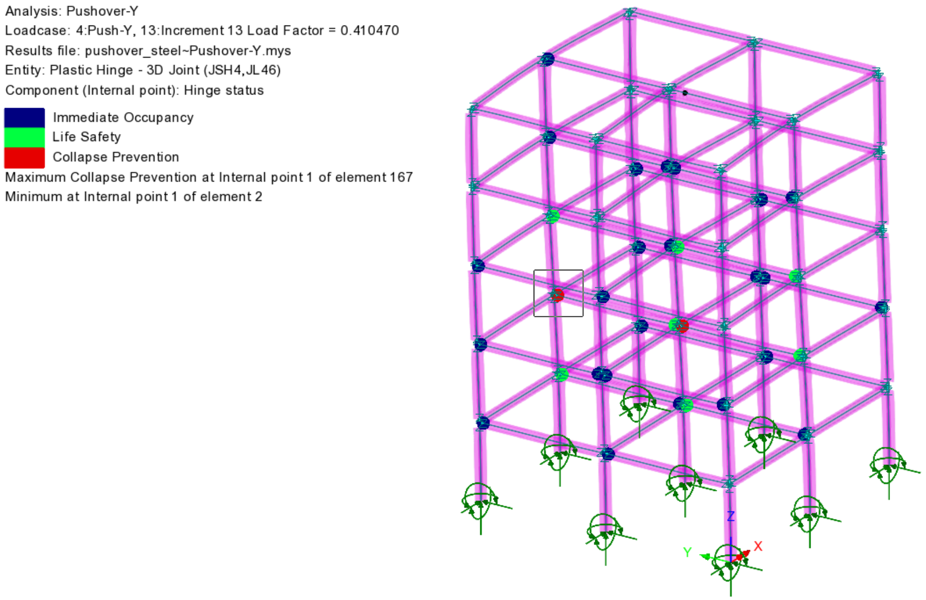
Tools
Animation wizard…

- Select **Load history** and press **Next.**
- In the 'Available' panel select **Push-Y** and press  to add all the load increments for this loadcase to the 'Included' panel, then press **Finish**.

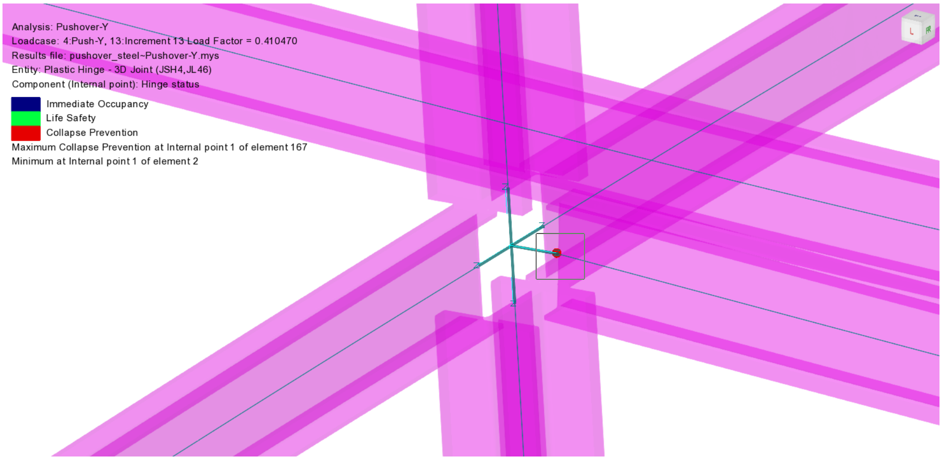


To see the location of the hinge

- Return to the model view window, which has load increment 13 still active.
-  Zoom-in to the joint coloured red to the left-rear of the frame, as shown on the following image.



The hinge can be seen to have formed at the beam connection with the column.

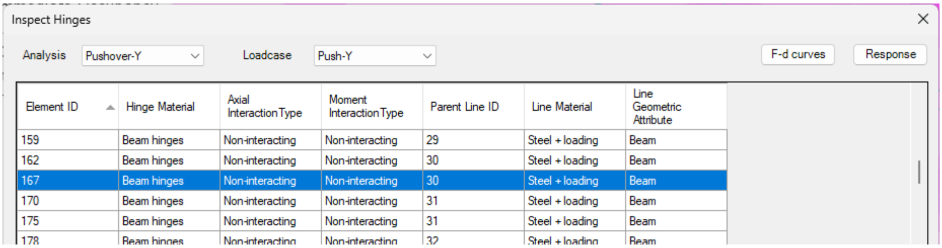


Note. Holding-down the E key (to select only Elements) and clicking and dragging a selection box around the joint highlighted with the red blob would confirm this is Element 167.

Hinge Response

The detailed response of the hinge can now be viewed:

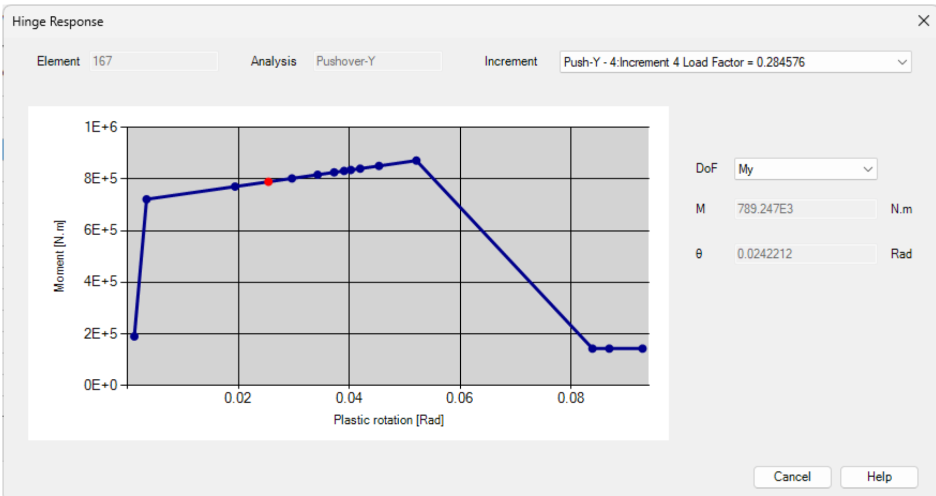
Tools
Inspect Hinges…



- Select Analysis **Pushover-Y** and loadcase **Push-Y**. Find **Element 167** in the list and press the **Response** button. If necessary sort the Element ID column to find it.

This displays the hinge response (deformations/rotations vs forces/moments) across the entire analysis.

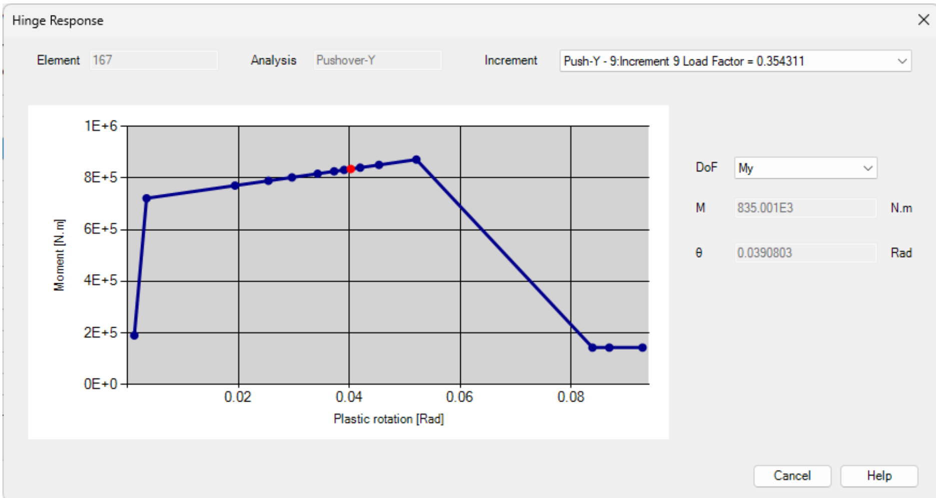
- On the Hinge Response dialog, in the Increment droplist, select **Increment 4**, the increment at which target displacement was determined. The location of this increment is shown with a red dot on the graph.

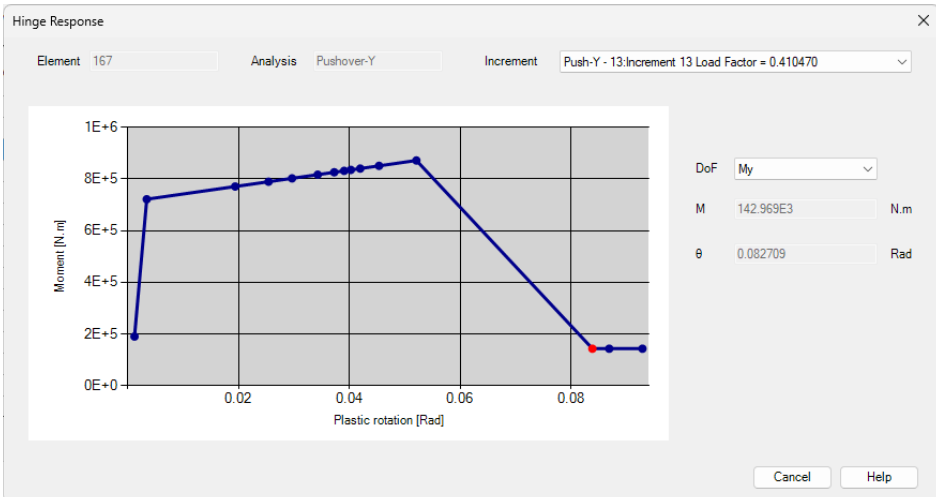


Note. Since this is a non-interacting hinge with only a degree of freedom of My, the DoF droplist only shows My as being available. For fully interacting hinges **Fx-My-Mz**, then Fx, My and Mz would be available.

Note. The greyed-out textboxes show the exact forces/moments and deformations/rotations that the hinge experiences at the given load increment. It should be ensured that the member can resist these loads.

- By changing the chosen increment to 9, and then 13, the location of the Life Safety and Collapse prevention cases can be seen.





Conclusions

This example of carrying out pushover analysis using LUSAS demonstrates the complete workflow required to evaluate the nonlinear seismic performance of a steel frame according to Eurocode 8. The study illustrates how eigenvalue and pushover analyses are defined, how plastic hinges are modelled and assigned, and how nonlinear controls are managed to ensure convergence. The resulting pushover curve enables the determination of the performance point (target displacement), while hinge status inspection and animation of the applied load factors provides insight into the structural response at critical stages.

The example highlights the effectiveness of pushover analysis in identifying critical hinge behaviour and evaluating seismic performance, offering engineers a reliable method for assessing structural resilience under earthquake loading.

References

EN 1998-1:2004 (2004) *Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings*, European Committee for Standardization, Brussels, Belgium.

Fajar, P. (2021) *The story of N2 method*. International Association for Earthquake Engineering

FEMA 356 (2000) *Prestandard and Commentary for Seismic Rehabilitation of Buildings*, Prepared by the American Society of Civil Engineers for the Federal Emergency Management Agency, Washington, D.C.