

Linear Buckling Analysis of a Flat Plate

For LUSAS version:	21.1
For software product(s):	Any Plus version.
With product option(s):	None.

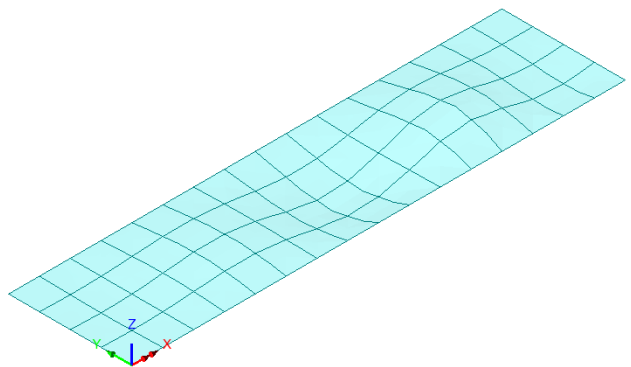
Description

This example determines the critical buckling load for a 2m x 0.5m rectangular panel of 1mm thickness subject to in-plane compressive loading.

Material properties for the panel are: Young's modulus 70E9 N/m², Poisson's ratio 0.3.

The panel is meshed using 64 thin shell elements and is simply supported on all sides. An in-plane compressive load of a total of 24N is applied to one of the short edges, parallel to the long sides.

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



Keywords

3D, Shell, Plate, Linear Buckling, Eigenvalue Buckling, Deformed Mesh, Print Results Wizard.

Associated Files

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



- ❑ **plate_modelling.lvb** This carries out the modelling of the example.

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.



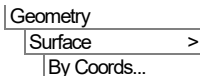
Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model


- Enter the file name as **plate**
- Use the default User-defined working folder.
- Ensure an analysis type of **Structural** is set.
- Select an Analysis Category of **3D**
- Set the model units to **N,m,kg,s,C**
- Ensure that timescale units are **Seconds**
- Select the model template **3D Beam/Shell**
- Enter the title as **Buckling of a flat plate**
- Click the **OK** button.



Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.



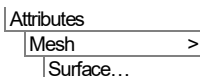
Feature Geometry

 Enter coordinates of $(0, 0)$, $(2, 0)$, $(2, 0.5)$ and $(0, 0.5)$ to define a Surface.


- Click the **OK** button.




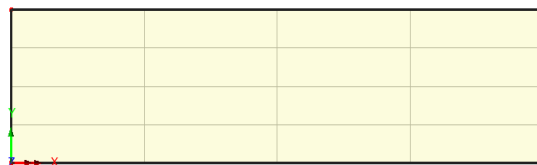
Meshing



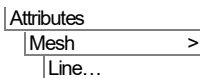
- Select **Thin shell**, **Quadrilateral**, elements with **Quadratic** interpolation.
- Enter the attribute name as **Thin Shell (QSL8)**.
- Click the **OK** button.


LUSAS will add the mesh dataset to the  Treeview.

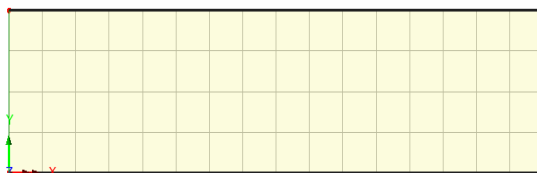
- Select the Surface of the plate.
- Drag and drop the Surface mesh attribute **Thin Shell** from the  Treeview onto the selected feature.



Note. If the number of divisions is not specified on the mesh dialog the default number of 4 divisions per Line will be used. In this example 4 divisions per Line is sufficient for the ends of the plate but 16 divisions are required on each of the sides.





- Define the number of element divisions to be **16**
- Enter an attribute name of **Null, div = 16** and click the **OK** button.
- Select the two long sides of the plate and drag and drop the Line mesh attribute **Null, div = 16** from the  Treeview onto the selected features.



Geometric Properties

- Specify a thickness of **0.001**.
- Enter the attribute name as **Plate Thickness**

(The eccentricity can be left blank, as it is not used in this analysis).

- Click the **OK** button to add the attribute to the  Treeview.
- With the Surface selected, drag and drop the geometry attribute **Plate Thickness** from the  Treeview onto the selected feature.

Assigned geometric attributes are visualised by default.





Select the fleshing on/off button to turn-off geometric property visualisation.


Material Properties

- Specify the Young's modulus as **70E9**
- Enter Poisson's ratio as **0.3**

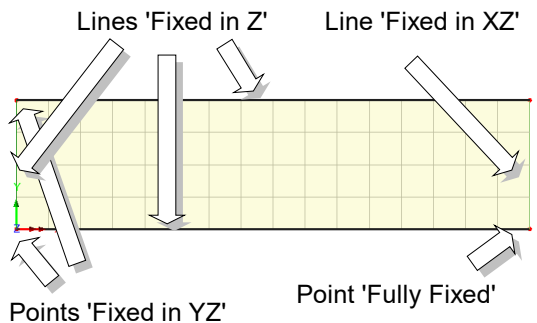
(Mass density can be left unspecified for Eigenvalue buckling analyses).




- Enter the attribute name as **Plate Material**
- Click the **OK** button to add the attribute to the  Treeview.
- With the Surface selected, drag and drop the material attribute **Plate Material** from the  Treeview onto the selected surface.

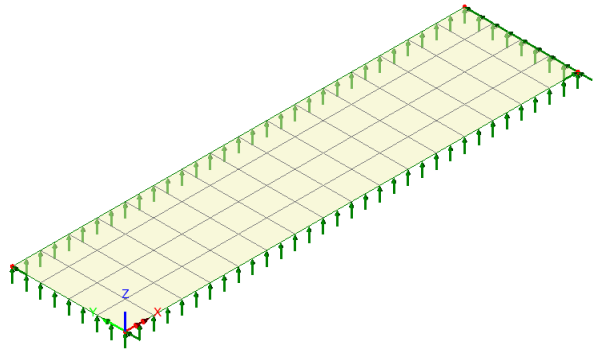
Supports

With the standard template, LUSAS provides the more common types of support by default. These can be seen in the  Treeview. Four support datasets are to be assigned to selected features of the model.

- For the 'Fixed in Z' support, select the top line, hold the **Shift** key, and select the left and bottom Lines as shown.



- Drag and drop the **Fixed in Z** support attribute from the  Treeview onto the selected Lines.
- Ensure that the supports are assigned to Lines for **Analysis 1** for **All loadcases** and click the **OK** button.
- Similarly, for each of the line and point features shown in the previous image drag and drop the relevant support attributes from the  Treeview to assign the required supports.
-  Use the Isometric button to rotate the model to this view.
- Check the position and type of supports on the model match those shown.




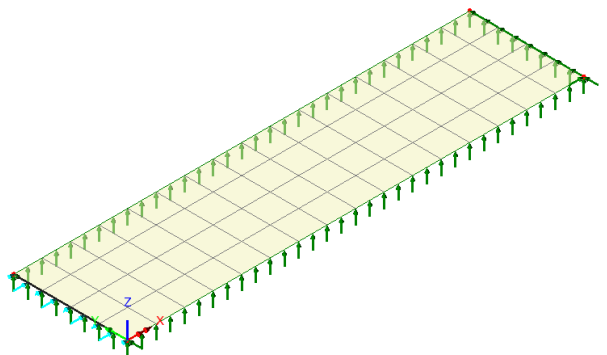
Loading

A global distributed load is to be applied to the left-hand end of the plate.

Attributes


Loading...

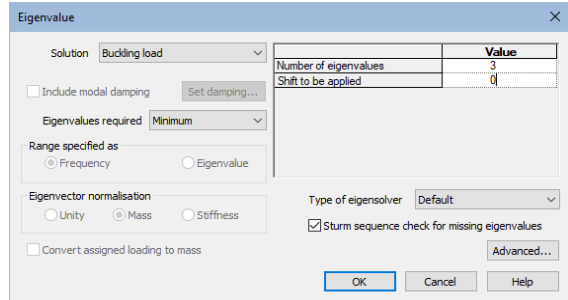
- Select the **Distributed loads** option, then **Global Distributed** and click **Next**
- Enter a **Total** load of **24** in the **X** direction.
- Enter the attribute name as **Distributed Load**.
- Click the **Finish** button.
- Select the left-hand edge of the plate and drag and drop the loading dataset **Distributed Load** from the  Treeview onto the selected Line.
- Click **OK** to assign the load to **Analysis 1** and **Loadcase 1**



Eigenvalue Analysis Control

Eigenvalue analysis control is defined as a loadcase property.

- In the  Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Eigenvalue** from the **Controls** menu.
- Select a **Buckling Load** solution for the **Minimum** number of eigenvalues.
- Enter the **Number of eigenvalues** required as **3**
- Enter the **Shift to be applied** as **0**
- Click the **OK** button to select the **Default** eigensolver.



Saving the model

File
Save



Save the model file.


Running the Analysis



Open the **Solve Now** dialog and press **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...

Eigenvalue results loadcases will be seen in the  Treeview.

In addition, these files will be created in the LUSASFiles32\`<model_name>` folder:



- ❑ **plate.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **plate.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...

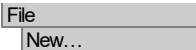
If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Select **No** to not view the output file. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported a command file is provided to enable you to re-create the model from scratch and run an analysis successfully.



- plate_modelling.lvb** carries out the modelling of the example.

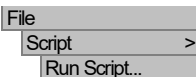


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **plate**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **2D Inplane** is set.
- Click the **OK** button.



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





- To recreate the model, select the file **plate_modelling.lvb** that was downloaded and placed in a folder of your choosing.




Solve the model to generate the results.

Viewing the Results

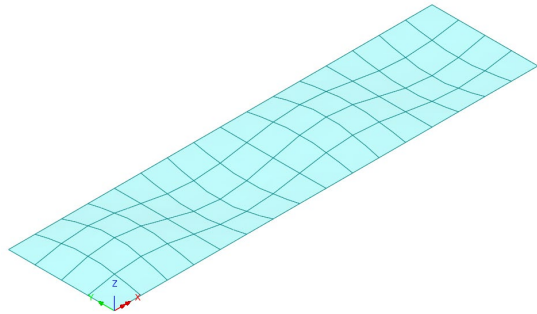
A deformed mesh plot will be seen in the View window. Loadcase results for each eigenvalue can be seen in the  Treeview. For eigenvalue analyses the first loadcase is set to be active by default.

- In the  Treeview, turn off the display of the **Mesh**, **Geometry** and **Attributes** layers.

Deformed Mesh Plot


- At the bottom of the  Treeview press the **Deformations...** button and see that **specify magnitude** is selected with a value of **6**.

If modelling in Imperial (US) units, 0.25 may be seen instead.



Changing the Results Loadcase

To view the second eigenmode:

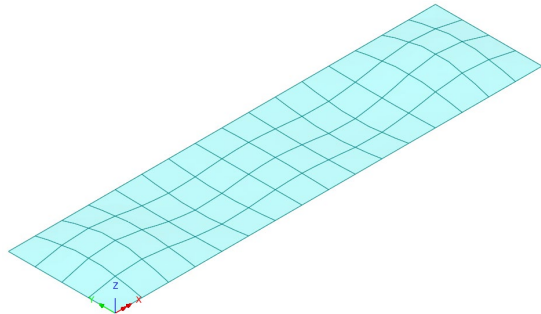
- In the  Treeview, in the eigenvalue results, right-click on **Mode 2 Load Factor = <value>** and select the **Set Active** option.

The second mode shape will be displayed.

The third eigenmode could be viewed in a similar manner.



Note. Mode shapes may be the opposite of those shown.

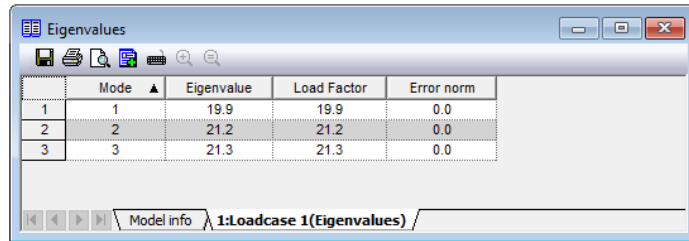


Printing the Buckling Load Factors

In an eigenvalue buckling analysis, the load factors are equivalent to the eigenvalues. Load factors are the values by which the applied load is factored to cause buckling in the respective modes. Eigenvalue results for the whole model can be displayed in the text window.

- Press the **Eigenvalues** radio button.
- From the Results loadcases drop-down menu select **All**, ensure that that **Eigenvalues (Load factor)** checkbox is selected and click the **OK** button.

The Eigenvalue and load factor results will be printed to the print results window.



The screenshot shows a software window titled "Eigenvalues" with a table of results. The table has five columns: Mode, Eigenvalue, Load Factor, and Error norm. The data is as follows:

	Mode ▲	Eigenvalue	Load Factor	Error norm
1	1	19.9	19.9	0.0
2	2	21.2	21.2	0.0
3	3	21.3	21.3	0.0

At the bottom of the window, there is a status bar with navigation arrows and the text "Model info 1: Loadcase 1 (Eigenvalues)".

Note that error norms may vary from those shown.

Calculating the Critical Buckling Load

The applied load (24N) must be multiplied by the first load factor (19.9) to give the value of loading which causes buckling in the first mode shape. The initial buckling load is therefore $24 \times 19.9 = 477.6$ N.



Note. An applied load of unity could be used in an eigenvalue analysis - in which case the eigenvalues produced would also represent the critical loads at which the structure would buckle. However, to prevent potential convergence problems with the analysis it is more usual to apply actual in-service loading and multiply the applied load by the eigenvalue to give the critical buckling load for each eigenvalue.

This completes the example.

