Arbitrary Section Property Calculation and Use

For LUSAS version:	17.0
For software product(s):	All (except LT versions)
With product option(s):	None.

Description

The section properties of an arbitrary shaped box section are to be computed and saved in a user library. Section geometry is supplied as a DXF file.

Units of N, mm, t, s, C ar**e** used.

Objectives
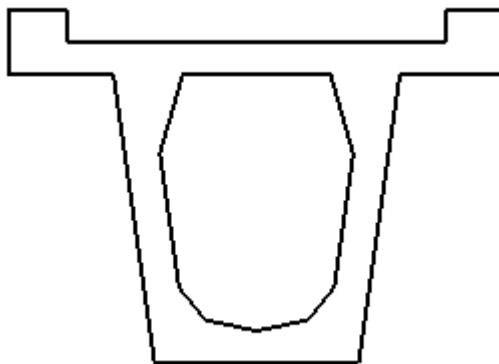
The required output from the analysis consists of:

❑ Section Properties of a box section

Keywords

2D, Inplane, Section Properties, Arbitrary Section, Box Section, Holes, Local Library, Server Library, Fibre locations.

Associated Files

❑ **box_section.dxf** DXF file containing geometry of section.

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 a File name of **box_section**

- Use the default User-defined working folder.

- Ensure an Analysis type of **Structural** is set.

- Select an Analysis Category of **2D Inplane**

- Set Model units of **N,mm,t,s,C**

- Leave the Timescale units as **Seconds**

- Select a Startup template of **None**.

- Ensure the Layout grid is set as **None**

- Enter a Title of **Box Section**

- Click the **OK** button.

Discussion

The arbitrary section property calculator within LUSAS Modeller computes the section properties of any open or closed section. Cross-sections are created either as a single regular or irregular surface, or as a group of surfaces (including holes). Fibre definitions (positions on the cross-section at which stresses or values can be plotted when viewing results) are also created for each drawn section at the time of section property calculation.

Caution. 2D cross-sections for section property calculation must be defined in the XY plane.
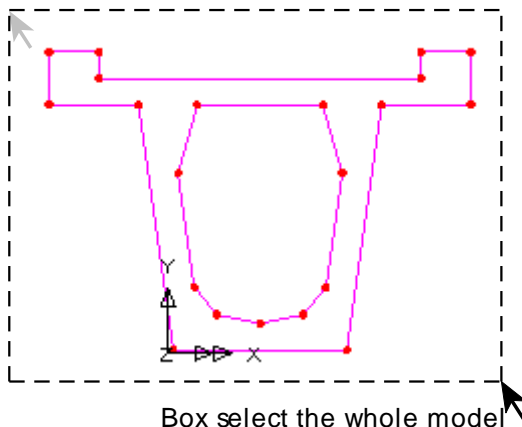
Feature Geometry

- Set the **Files of type** drop-down menu to **DXF Files (*.dxf)**

- Locate the **box_section.dxf** file in the **\<Lusas Installation Folder>\Examples\Modeller** directory and click the **Import** button to read in the DXF file and create the cross section geometry as shown below.

Defining holes within a section

This is done by first defining a surface that represents the total extent of the box-section, then defining a surface that represents the void or hole, and then selecting both the surrounding and inner surface to create the hole in the bounding surface.

- Drag a box around the whole model
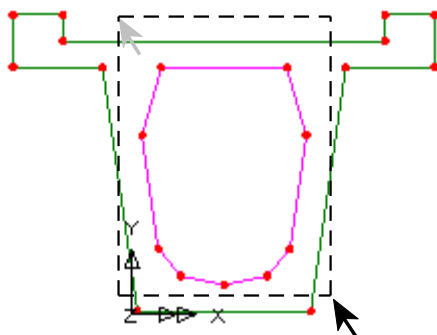
Box select the whole model

Click the **OK** button to create a **General Surface** (for the outer surface) from the selected Lines.

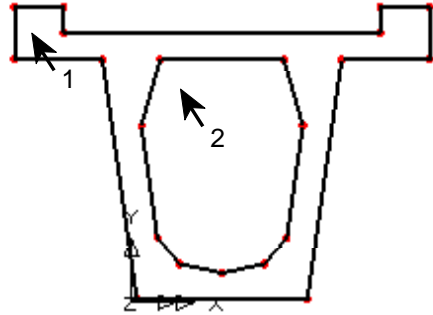- Now, drag a box around the only the lines defining the void.

Click the **OK** button to create a **General Surface** (for the inner surface) from the selected Lines.
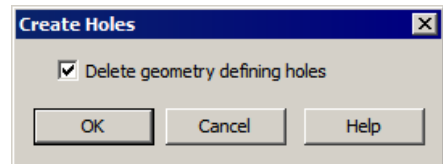
Box select the lines defining the void

- Select the outer Surface of the box-section, then hold the **Shift** key down, and select the inner Surface representing the void.

Select these two surfaces

Geometry	
Surface	>
Holes	>
Create	

- On the Create Holes dialog, ensure the option **Delete geometry defining holes** is selected, and click **OK**.

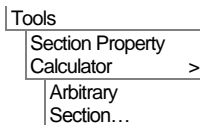A new singular Surface will be created, containing a void. This can be seen by clicking inside the outer perimeter of the box section and seeing that all lines are selected.

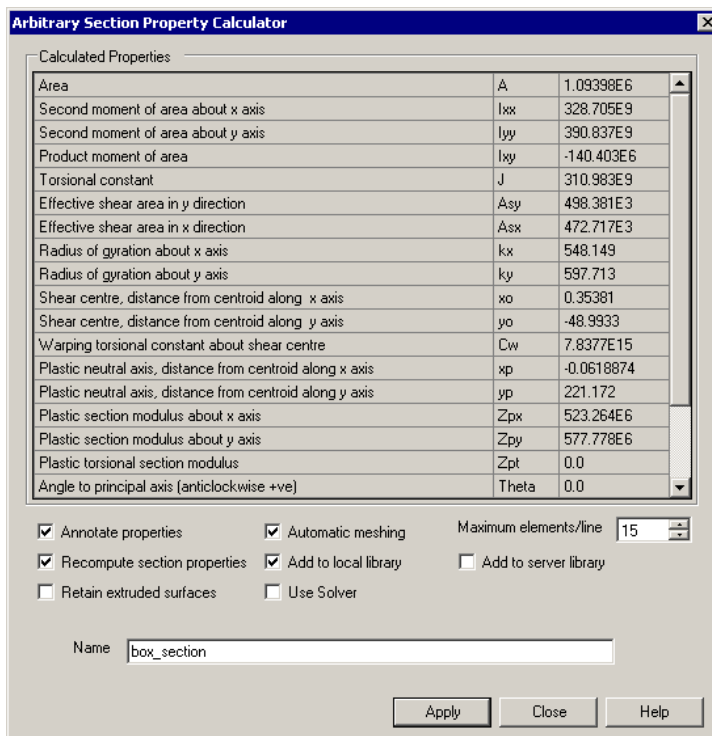| File	
Save...	

Save the cross-section model for any future modifications or for use with the definition of any additional fibre locations.

The section properties can now be calculated as follows:

Calculating Section Properties

• Select the **Add to local library** option.

• Click the **Apply** button, and after a short wait the calculated section properties will be displayed in the grid at the top of the dialog.

Arbitrary Section Property Calculator

Calculated Properties

Area	A	1.09398E6
Second moment of area about x axis	Ixx	328.705E9
Second moment of area about y axis	Iyy	390.837E9
Product moment of area	Ixy	-140.403E6
Torsional constant	J	310.983E9
Effective shear area in y direction	Asy	498.381E3
Effective shear area in x direction	Asx	472.717E3
Radius of gyration about x axis	kx	548.149
Radius of gyration about y axis	ky	597.713
Shear centre, distance from centroid along x axis	xo	0.35381
Shear centre, distance from centroid along y axis	yo	-48.9933
Warping torsional constant about shear centre	Cw	7.8377E15
Plastic neutral axis, distance from centroid along x axis	xp	-0.0618874
Plastic neutral axis, distance from centroid along y axis	yp	221.172
Plastic section modulus about x axis	Zpx	523.264E6
Plastic section modulus about y axis	Zpy	577.778E6
Plastic torsional section modulus	Zpt	0.0
Angle to principal axis (anticlockwise +ve)	Theta	0.0

☑ Annotate properties   ☑ Automatic meshing   Maximum elements/line 15

☑ Recompute section properties ☑ Add to local library   ☐ Add to server library

☐ Retain extruded surfaces   ☐ Use Solver

Name box_section

[Apply]   [Close]   [Help]

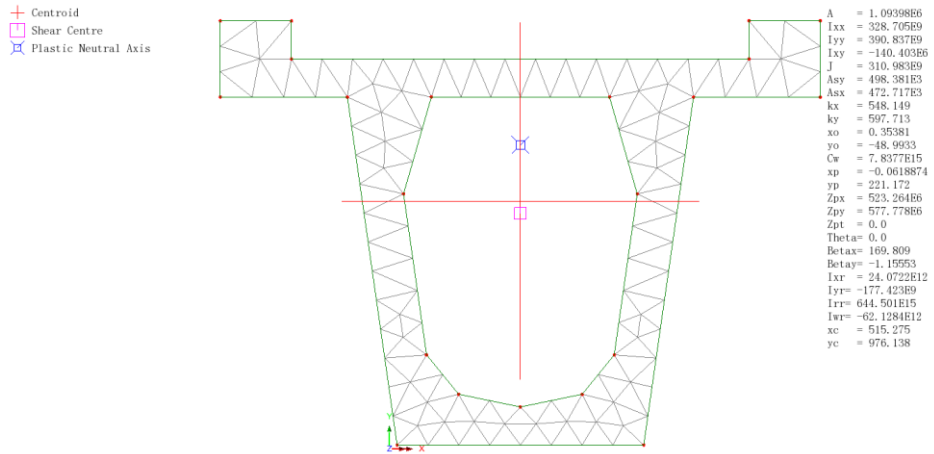• Click **Cancel** to close the dialog.

Note. There is no need to compute the section properties in the units they are to be used in the analysis model because units conversion is carried out when the section properties are extracted from the section library if it is found to be required.

Note. Section properties may be added to local (user) or server (all users) section property libraries by selecting the appropriate option(s) prior to selecting the **Apply** button. Alternatively you can deselect the 'Recompute section properties' option after selecting the 'Add local library' or 'Add server library' options. By default the model name is entered as the Section Name. This can be modified if required.

Notes on the automatic meshing used

The mesh used to compute the properties of each of the surfaces (and the holes) is displayed in the graphics window. Centroid, Shear Centre and Plastic Neutral Axis symbols and locations are also shown.
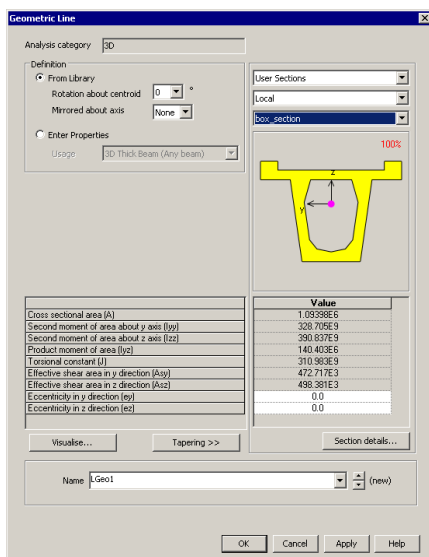

By default an element size is selected which will assign 15 elements to the longest side and a minimum of 2 elements is applied to the shorter sides. This mesh may be adjusted by deselecting the automatic mesh check box and changing the mesh size in the Treeview. Alternatively, the maximum element on the longest side may be adjusted by changing the 'Max elements/line' option as required prior to selecting the Apply button. As with all finite element models the more elements used the more accurate the results but the slower the calculation. A good compromise of 2 elements across all thin sections has been found to provide reasonable results without using excessive computation time.

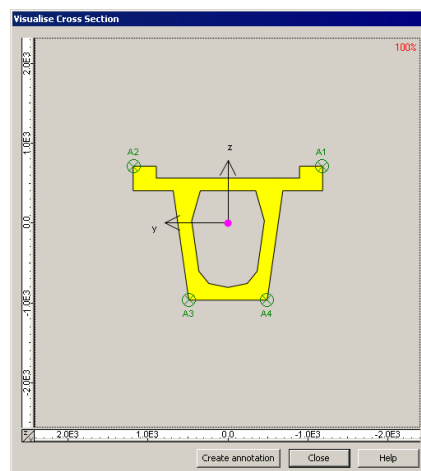## Viewing and using sections in the user library

Create a new model called **Test**, select an analysis category of **3D**, and select units of **N,mm,t,s,C** from the drop down lists. Click **OK**.

File
New…

Attributes
Geometric   >
Section library…

- Select **User Sections** from the top-right drop-down on the dialog

- In the Local library, browse for, and select the **box_section** name. The basic section will be visualised. Note that eccentricity values can be defined here for sections, when appropriate.

- Press the **Section details** button and the fibre definitions will be seen. Click **Close**

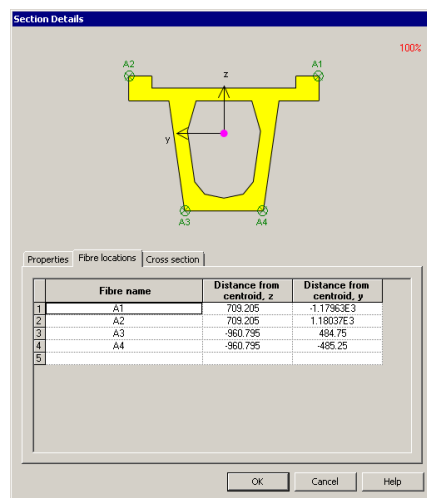**Visualisation of local library item**   **Visualisation of fibre locations**

Fibre locations are used to calculate beam stress results for plotting force and moment diagrams. LUSAS automatically creates one for every extreme point on a section.

- On the Geometric Line dialog, press the **Section details** button.

- Select the **Fibre locations** tab. The calculated extreme fibre locations can be seen.

Note that additional fibre locations can be added manually to the library item by clicking in the Fibre locations area and pressing the **TAB** key until a new line is created, and then entering the coordinate of the fibre location required.

- Press **OK** to save changes and return to the Geometric Line dialog.

Tip. If additional fibre locations are required on the section use Modeller to obtain the properties of any additional points on the saved model and enter the point coordinates

in the manner described above. Note that if invalid coordinates are used the fibre location will be added to the nearest point on the outer perimeter of the section.

- Finally, to add the local library section to the Attributes Treeview (for subsequent assignment to selected lines representing beams in the model) press the **OK** button.

This completes the example.