# Arbitrary Section Property Calculation and Use

For LUSAS version:	19.0
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 are 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 <u>must</u> be defined in the XY plane.

# Feature Geometry

File Import...

- 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



G	eometry	
	Surface	>
	By Lines	

# 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.

G	eometry	
	Surface	>
	By Lines	

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 boxsection, then hold the **Shift** key down, and select the inner Surface representing the void.



Select the two surfaces

G	iec	ometry	
	S	urface	>
		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.

 
 Section Property Calculator
 >

 Arbitrary Section...
 >

Area	A	1.09398E6	
Second moment of area about x axis	box	328.705E9	
Second moment of area about y axis	lyy	390.837E9	
Product moment of area	bxy	-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	1
Radius of gyration about y axis	ky	597.713	
Shear centre, distance from centroid along x axis	хо	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	хр	-0.050769	
Plastic neutral axis, distance from centroid along y axis	ур	221.17	
Plastic section modulus about x axis	Zpx	519.019E6	
Plastic section modulus about y axis	Zpy	578.144E6	
Plastic torsional section modulus	Zpt	579.066E6	
Angle to principal axis (anticlockwise +ve)	Theta	0.0	
Annotate properties     A	Maximum eler	nents/line 15 ver library	
Name box_section			

• Click **Close** 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 conversion of units 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**.

Attributes
Geometric >
Section library...

File

New..

- 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 Visualise button and the fibre locations will be seen. Click Close.

	×	Visualise Cross Section
nalysis category 30 Definition © From Library Rotation about centroid 0 v ° Mirrored about cavis None v	Loser Sections   Local  box_section  v	10 87 87
Beter Properties Usage ① Thick Beam (Any beam) *		
None   Cons sectional area (h)  Second moment of area about y axis (hy)  Second moment of area about y axis (hy)  Product moment of area (hy)  Throad contart(1)  Effective about area in y direction (Apa)  Effective about area in y direction (Apa)	Volue           10539856           328,70569           398,8759           140,40356           310,93859           472,71753           498,38153	
Usualse Tapering >>	0.0 0.0 Section details	[5]         2 (63         - 1 (63         - 2 (63           Create annotation         Cose         Help

Visualisation of local library item



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.



**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

, de la companya de l			
roperties Fibre locations Cross section			
roperties Fibre locations Cross section	es 🔽 Plastic	: properties	
roperties Fibre locations Cross section	es 👽 Plastic	properties	
roperties Fibre locations Cross section V Standard properties V Additional propertie Description	es 🔽 Plastic	; properties Value box_section	
Type Type Type Type Type Type Type Type	es V Plastic	properties Value box_section	
roperties [Eine locations] Cross section] 2 Standard properties 2 Additional propertie Description Type Cross sectional area Second serverated of sea allow if y area	es V Plastic	properties Value box_section 1.09398E6 328.705E9	
roperties Etime locations Cross section Standard properties Additional propertie Description Top Cross sectional area Cross sec	es V Plastic	z properties Value box_section 1.09398E6 328.705E9 390.827E9	
Toperties Fibre locations Cross section  Standard properties Additional propertie  Description Type Cross sectional area Second moment of area about y axis Second moment of area	es V Plastic A hy lizz byz	z properties Value box_section 1.09398E6 328.705E9 390.637E9 140.403E6	
roperties Ethre locations Cross section Standard properties Additional propertie Description Type Crossectional area Crossectional area Crossect	es V Plastic	z properties Value box_section 1.09398E6 328.705E9 390.837E9 140.403E6 310.983E9	
Troper ties Fibre locations Cross section  Standard properties Additional propertie  Description Type Cross sectional area Second moment of area about y axis Second moment of area about y axis Tonisonal constant Tensional constant Tensional constant	A by lzz byz J Asv	z properties Value box_section 1.09398E6 328.705E9 390.837E9 140.403E6 310.983E9 472.717E3	
Toper Hes Elize locations Cross section  Standard properties Additional propertie  Second moment of area about y axis  Second moment of area about y axis  Product moment of area about y axis  Product moment of area about y axis  Froduct area area in y detection  Effective shear area in y detection	A A by lzz byz J Asy Asz	properties Value box_section 1.09398E6 328.705E9 390.837E9 140.403E6 310.983E9 472.717E3 498.381E3	
Toperties Fibre locations Cross section  Standard properties Additional propertie  Description Type Cross sectional area Second moment of area about y axis Second moment of area about y axis Torsional constart Finduse shear area in y direction Effective shear area in y direction Effective shear area in y direction Effective shear area in y direction	A by lzz byz J Asy Asz ey	rproperties Value box_section 1.09398E6 328.70E9 390.837E9 140.403E6 310.983E9 472.717E3 498.381E3 0.0	
Toper Hes Etime locations Cross section  Standard properties Additional propertie  Second moment of area about y axis Technologic and the second of the second	es ♥ Plastic A byv lzz byz J Asy Asz ey ez	2 properties Value box_section 1.09398E6 328.705E9 390.837E9 390.837E9 310.983E9 310.983E9 472.777E3 498.381E3 0.0 0.0	

will be added to the nearest point on the outer perimeter of the section.

- Press **OK** to save changes and return to the Geometric Line dialog.
- Finally, to add the local library section to the Attributes here assignment to selected lines representing beams in the model) press the **OK** button.

This completes the example.